



# **Development of a NURBS-Based Computational Method for the VAST Finite Element Program**

## *Phase 2 – Final Report*

*Lei Jiang and Eric Teng  
Martec Limited*

*Martec Limited  
Suite 400, 1888 Brunswick St.  
Halifax, Nova Scotia B3J 3J8*

*Project Manager: Dave Brennan, 902-425-5101 ext 226*

*Contract Number: W7707-06-3597/001/HAL Task 09*

*Contract Scientific Authority: Dave Stredulinsky, 902-426-3100 ext 352*

## **Defence R&D Canada – Atlantic**

Contract Report  
DRDC Atlantic CR 2008-118  
October 2008

This page intentionally left blank.

# **Development of a NURBS-Based Computational Method for the VAST Finite Element Program**

Phase 2 – Final Report

Lei Jiang and Eric Teng

Martec Limited  
400-1888 Brunswick Street  
Halifax, Nova Scotia B3J 3J8

Project Manager: Dave Brennan, 902-425-5101 ext 226

Contract number: W7707-06-3597/001/HAL Task 09

Contract Scientific Authority: Dave Stredulinsky, 902-426-3100 ext 352

**Defence R&D Canada – Atlantic**

**Contract Report**

DRDC Atlantic CR 2008-118

October 2008

Authors

---

Lei Jiang

Approved by

*Original signed by N.G. Pegg*

---

N.G. Pegg

Head/Warship Performance

Approved for release by

*Original signed by Ron Kuwahara for*

---

Calvin Hyatt

Chair/Document Review Panel

The scientific or technical validity of this Contract Report is entirely the responsibility of the contractor and the contents do not necessarily have the approval or endorsement of Defence R&D Canada

## Abstract

---

This report presents a recent project for further verification and enhancement of a recently developed NURBS-based computational method in VAST. This method has the potential for close integration of engineering analysis capability with Computer Aided Design (CAD) systems. It also has the advantages of preserving exact geometry of structures to be analyzed and providing a convenient means for mesh refinement. In a previous project, the NURBS-based formulation was implemented into the VAST program and verified for a wide variety of engineering analyses, including linear static, eigenvalue and nonlinear analyses involving plastic post-collapse responses of thin shell structures. In this work, additional verification of the NURBS-based element is performed and the numerical problems observed in the previous study were also investigated and resolved. A post-processing capability for the NURBS-based element is developed and the possibility for applying this element for solving practical engineering problems is also explored. This study confirmed advantages of NURBS-based elements from the computational point of view. An automatic model generator for NURBS-based elements still needs to be developed in the future.

## Résumé

---

Ce rapport présente un projet récent de vérification additionnelle et d'amélioration d'une méthode de calcul basée sur NURBS qui est intégrée à VAST et qui pourrait potentiellement permettre une intégration plus étroite des capacités d'analyse d'ingénierie à des systèmes de conception assistée par ordinateur (CAO). Cette méthode présente aussi l'avantage de préserver la géométrie exacte des structures analysées et elle constitue un moyen pratique de raffinement des maillages. Dans le cadre d'un projet précédent, la formulation fondée sur NURBS avait été mise en oeuvre dans le programme VAST et vérifiée avec une vaste gamme d'analyses en ingénierie, notamment des analyses statiques linéaires, de valeurs propres et non linéaires portant sur les réactions plastiques après effondrement de structures à enveloppe mince. Au cours de ces travaux, une vérification plus poussée de l'élément basé sur NURBS a été effectuée et les problèmes numériques observés au cours de l'étude précédente ont aussi été étudiés et résolus. Une capacité de post-traitement est en cours de développement et on explore la possibilité d'appliquer cet élément à la résolution de problèmes d'ingénierie concrets. Cette étude a confirmé les avantages de l'élément basé sur NURBS du point de vue computationnel. Par ailleurs, il est nécessaire de développer un générateur de modèles automatisé pour des éléments basés sur NURBS.

This page intentionally left blank.

## Executive summary

---

### Development of a NURBS-based Computational Method for the VAST Finite Element Program: Phase 2 – Final Report

Lei Jiang, Eric Teng; DRDC Atlantic CR 2008-118; Defence R&D Canada - Atlantic;  
October 2008

**Introduction:** A new numerical method for solving problems governed by partial differential equations, such as solids and structures, has been developed recently, taking inspiration from Computer Aided Design (CAD). Although this method has many features in common with the finite element method and some features in common with the meshless methods, it is more geometrically based and provides a potential for a close integration with CAD systems. This numerical formulation is based on NURBS (Non-Uniform Rational B-Splines), a standard technology employed in CAD systems. Using this method, exact geometric representations of the solids and structures are maintained for any level of mesh refinement.

**Results:** The NURBS-based computational method was implemented into the VAST finite element program in a previous contract as a special element type, similar to the conventional solid element. Under the present contract, further verification of the NURBS-based element was performed and the performance of the NURBS-based and traditional 20-noded solid elements was extensively compared. The previously observed convergence difficulty of the NURBS-based element was investigated and resolved. To improve user friendliness of the NURBS-based element, a post-processing capability was developed. Finally, a NURBS-based finite element model of a simplified ship hull structural component consisting of a hull plate and intersecting truncated transverse bulkhead and two longitudinal stiffeners was created and analyzed. The results indicated that to achieve the same level of accuracy, the size of the NURBS-based finite element model was considerably smaller than that of the traditional 20-noded solid element model, resulting in a much shorter computation time.

**Significance:** The present study confirmed the advantages of the NURBS-based element formulation from the computational point of view. This formulation provides an exact representation of the problem geometry and a convenient approach for further mesh refinement through insertion of knots and control points. The NURBS-based element is now working reliably for all types of analyses and its potential for analyses of ship structural components has been demonstrated.

**Future plans:** Significant developments are still required before it can be utilized in practical engineering analyses of complete ship structures and structural connection details. A pre-processing capability is needed to take the CAD model and transform it into an initial mesh or control net of NURBS-based solid finite elements. In addition, the possibility for developing NURBS-based shell elements should be explored.

## Sommaire

---

### Développement d'une méthode computationnelle basée sur NURBS pour le programme de calcul par éléments finis VAST : Phase 2 – Rapport final

Lei Jiang, Eric Teng; RDDC Atlantique CR 2008-118; R&D pour la défense Canada  
- Atlantique; October 2008

**Introduction :** Une nouvelle méthode numérique de résolution de problèmes fondée sur des équations différentielles partielles, comme celles appliquées aux solides et aux structures, a été développée récemment en s'inspirant de la conception assistée par ordinateur (CAO). Bien que cette méthode partage plusieurs caractéristiques avec la méthode utilisant des éléments finis et quelques caractéristiques avec les méthodes sans maillage, elle est plutôt basée sur la géométrie et elle présente le potentiel d'une intégration étroite avec les systèmes de CAO. Cette formulation mathématique est basée sur NURBS (courbes B-splines rationnelles non uniformes), une technologie standard utilisée dans les systèmes de CAO. Au moyen de ce système, il est possible de conserver des représentations géométriques exactes de solides et de structures pour tout degré de raffinement des maillages et aucune communication ultérieure avec la description CAO n'est requise au cours du processus de raffinement du maillage au moyen d'éléments finis.

**Résultats :** La méthode de calcul basée sur NURBS a été mise en oeuvre dans le programme de calcul par éléments finis VAST dans le cadre d'un contrat précédent sous forme d'un type d'élément spécial, semblable à un élément solide classique. Dans le cadre du présent contrat, on a effectué des vérifications plus poussées de l'élément basé sur NURBS et on a effectué une comparaison poussée des éléments solides basés sur NURBS et des solides classiques à 20 noeuds. Le problème de convergence touchant l'élément basé sur NURBS, relevé précédemment, a été étudié et résolu. Afin d'améliorer la convivialité de l'élément basé sur NURBS, une capacité de post-traitement a été développée. Finalement, on a créé et analysé un modèle d'élément fini basé sur NURBS représentant une composante structurale de coque de navire simplifiée composée d'un bordé de carène (tôle de coque), d'une cloison transversale tronquée en intersection et de deux lisses longitudinales, a été créée et analysée. Les résultats obtenus indiquent que, pour obtenir le même degré de précision, la taille du modèle à éléments finis basé sur NURBS était considérablement plus petite que celle du modèle à élément solide à 20 noeuds classique, ce qui réduit considérablement le temps de calcul.

**Importance :** L'étude actuelle a confirmé les avantages de la formulation d'éléments basée sur NURBS du point de vue de la quantité de calcul requise. Cette formulation fournit une représentation exacte de la géométrie du problème et elle constitue une approche commode pour raffiner le maillage en insérant des noeuds et des points de contrôle. L'élément basé sur NURBS fonctionne de manière fiable pour tous les types d'analyse et nous avons démontré son potentiel dans le cas de l'analyse des composantes structurelles des navires.

**Plans futurs :** Des travaux de développement additionnels sont encore requis afin que cette méthode puisse être appliquée à des analyses d'ingénierie pratiques de structures complètes de navires et d'analyses des détails des assemblages structuraux. Une fonction de prétraitement est requise afin de pouvoir transformer les modèles de CAO en des maillages initiaux ou des réseaux de contrôle d'éléments finis solides basés sur NURBS. En outre, il serait nécessaire d'explorer la possibilité de développer des éléments d'enveloppe basés sur NURBS.



**TABLE OF CONTENTS**

---

1.0 INTRODUCTION .....1

2.0 DEVELOPMENT OF A POST-PROCESSING CAPABILITY FOR THE NURBS-BASED SOLID  
ELEMENT .....3

3.0 CONVERGENCE STUDY OF THE NURBS-BASED SOLID ELEMENT FOR LINEAR STATIC  
ANALYSIS .....11

4.0 AN INVESTIGATION OF ACCURACY OF THE NURBS-BASED SOLID ELEMENT FOR  
BUCKLING ANALYSIS.....15

5.0 COMPARISON OF COMPUTATIONAL EFFICIENCY OF THE NURBS-BASED AND  
TRADITIONAL SOLID ELEMENTS FOR SHIP STRUCTURE ANALYSES .....30

6.0 CONCLUSIONS.....41

7.0 REFERENCES .....43

## LIST OF FIGURES

FIGURE 2.1: CONTROL NET OF NURBS-BASED SOLID ELEMENT MODEL OF A CYLINDRICAL SHELL .....	5
FIGURE 2.2: A 20-NODED SOLID ELEMENT MODEL OF THE CYLINDRICAL SHELL GENERATED BY THE POST-PROCESSOR.....	5
FIGURE 2.3: DISPLACEMENT CONTOUR GENERATED USING THE POST-PROCESSING CAPABILITY FOR GEOMETRICALLY NONLINEAR ANALYSIS OF A HINGED CYLINDRICAL SHELL .....	6
FIGURE 2.4: CONTROL NET OF NURBS-BASED SOLID ELEMENT MODEL OF A SPHERICAL SHELL .....	7
FIGURE 2.5: A 20-NODED SOLID ELEMENT MODEL OF THE SPHERICAL SHELL GENERATED BY THE POST-PROCESSOR.....	7
FIGURE 2.6: DISPLACEMENT CONTOUR GENERATED USING THE POST-PROCESSING CAPABILITY FOR GEOMETRICALLY NONLINEAR ANALYSIS OF A HINGED SPHERICAL SHELL.....	8
FIGURE 2.7: STRESS CONTOUR PLOTS GENERATED USING THE POST-PROCESSOR AT A TYPICAL LOAD STEP IN NONLINEAR ANALYSIS OF A HINGED SPHERICAL SHELL.....	9
FIGURE 2.8: PLASTIC STRAIN CONTOUR PLOTS GENERATED USING THE POST-PROCESSOR AT A TYPICAL LOAD STEP IN NONLINEAR ANALYSIS OF A HINGED SPHERICAL SHELL .....	10
FIGURE 3.1: TYPICAL 20-NODED SOLID ELEMENT MESHES UTILIZED IN CONVERGENCE STUDY .....	13
FIGURE 3.2: COMPARISON OF CENTRE DEFLECTIONS PREDICTED BY NURBS-BASED SOLID ELEMENT AND 20-NODED SOLID ELEMENT WITH FULL NUMERICAL INTEGRATION.....	14
FIGURE 3.3: COMPARISON OF CENTRE DEFLECTIONS PREDICTED BY NURBS-BASED SOLID ELEMENT AND 20-NODED SOLID ELEMENT WITH REDUCED NUMERICAL INTEGRATION.....	14
FIGURE 4.1: PROBLEM DEFINITION FOR BUCKLING ANALYSES OF A HINGED CYLINDRICAL SHELL SUBJECTED TO A CENTRE POINT LOAD.....	18
FIGURE 4.2: CONTROL NET OF A NURBS-BASED SOLID ELEMENT MODEL REPRESENTING ONE QUARTER OF THE CYLINDRICAL SHELL WITH BOUNDARY CONDITIONS AND EXTERNAL LOAD .....	19
FIGURE 4.3: BUCKLING LOADS OBTAINED BY NURBS-BASED SOLID ELEMENTS OF VARIOUS ORDERS AND 20-NODED SOLID ELEMENTS PRESENTED IN PREVIOUS CONTRACT REPORTED [1] .....	20
FIGURE 4.4: BUCKLING MODE SHAPES FOR MODE 2 AND 5 PREDICTED BY THE 20-NODED SOLID ELEMENTS .....	21
FIGURE 4.5: BUCKLING MODE SHAPES FOR MODE 2 AND 5 PREDICTED BY NURBS-BASED SOLID ELEMENT MODEL OF ORDER $3 \times 3 \times 2$ USING $3 \times 3 \times 2$ NUMERICAL INTEGRATION.....	22
FIGURE 4.6: BUCKLING MODE SHAPES FOR MODE 2 AND 5 PREDICTED BY NURBS-BASED SOLID ELEMENT MODEL OF ORDER $5 \times 5 \times 2$ USING $4 \times 4 \times 3$ NUMERICAL INTEGRATION.....	23
FIGURE 4.7: PROBLEM DEFINITION FOR BUCKLING OF A SQUARE PLATE SUBJECTED TO BIAXIAL COMPRESSION AND THE CONTROL NET OF A NURBS-BASED SOLID ELEMENT MODEL OF ORDER $5 \times 5 \times 2$ REPRESENTING ONE QUARTER OF THE PLATE (SHADED AREA) .....	24
FIGURE 4.8: BUCKLING MODES 4 AND 9 OF THE BI-AXIALLY LOADED PLATE PREDICTED BY NURBS-BASED SOLID ELEMENT MODEL OF ORDER $5 \times 5 \times 2$ USING $8 \times 8 \times 3$ NUMERICAL INTEGRATION .....	25
FIGURE 4.9: BUCKLING LOAD FACTORS OBTAINED BY NURBS-BASED SOLID ELEMENTS OF VARIOUS POLYNOMIAL ORDERS USING PROPOSED DEFAULT ORDERS OF NUMERICAL INTEGRATION .....	26
FIGURE 4.10: BUCKLING MODES 2 AND 5 OF THE CYLINDRICAL SHELL PREDICTED BY NURBS-BASED SOLID ELEMENT MODEL OF ORDER $5 \times 5 \times 2$ USING $5 \times 5 \times 2$ NUMERICAL INTEGRATION .....	27
FIGURE 4.11: LINEAR STATIC SOLUTIONS OBTAINED USING NURBS-BASED SOLID ELEMENT MODEL OF POLYNOMIAL ORDER $5 \times 5 \times 2$ USING $4 \times 4 \times 3$ NUMERICAL INTEGRATION .....	28
FIGURE 4.12: LINEAR STATIC SOLUTIONS OBTAINED USING NURBS-BASED SOLID ELEMENT MODEL OF POLYNOMIAL ORDER $5 \times 5 \times 2$ USING $5 \times 5 \times 2$ NUMERICAL INTEGRATION .....	29
FIGURE 5.1: SURFACE MODEL OF A STIFFENED PLATE STRUCTURAL COMPONENT .....	34
FIGURE 5.2: ISOGEOMETRIC NURBS MODEL OF THE STIFFENED PLATE STRUCTURAL COMPONENT .....	34
FIGURE 5.3: CONTROL NET OF THE NURBS-BASED SOLID ELEMENT MODEL OF THE STIFFENED PLATE STRUCTURAL COMPONENT SHOWN WITH BOUNDARY CONDITIONS AND EXTERNAL LOAD .....	35
FIGURE 5.4: UNDEFORMED CONTROL NET AND DEFORMED CONFIGURATION OF A NURBS-BASED SOLID ELEMENT MODEL OF THE STIFFENED PLATE STRUCTURAL COMPONENT .....	36
FIGURE 5.5: UNDEFORMED AND DEFORMED MESHES OF THE COARSE MESH OF 20-NODED SOLID ELEMENT OF THE STIFFENED PLATE STRUCTURAL COMPONENT.....	37
FIGURE 5.6: UNDEFORMED AND DEFORMED MESHES OF THE FINE MESH OF 20-NODED SOLID ELEMENT OF THE STIFFENED PLATE STRUCTURAL COMPONENT.....	38

FIGURE 5.7: STRESS CONTOUR PREDICTED BY THE NURBS-BASED SOLID ELEMENT MODEL OF THE STIFFENED PLATE STRUCTURAL COMPONENT.....	39
FIGURE 5.8: STRESS CONTOUR PREDICTED BY THE COARSE (TOP) AND FINE (BOTTOM) SOLID ELEMENT MODELS OF THE STIFFENED PLATE STRUCTURAL COMPONENT .....	40

**LIST OF TABLES**

TABLE 3.1: RESULTS OF CONVERGENCE STUDY USING 20-NODED SOLID ELEMENT - CENTRE DEFLECTION OF A HINGED CYLINDRICAL SHELL SUBJECTED TO A POINT LOAD.....	12
TABLE 4.1: COMPARISON OF CRITICAL STRESSES FOR BUCKLING OF A BI-AXIALLY LOADED SIMPLY SUPPORTED SQUARE OBTAINED BY A NURBS-BASED SOLID ELEMENT MODEL USING DIFFERENT ORDERS OF NUMERICAL INTEGRATION. ....	17
TABLE 4.2: COMPARISON OF CRITICAL STRESSES FOR BUCKLING OF A BI-AXIALLY LOADED SIMPLY SUPPORTED SQUARE OBTAINED BY DIFFERENT NURBS-BASED SOLID ELEMENT MODELS USING THE DEFAULT NUMERICAL INTEGRATION. ....	17
TABLE 5.1: COMPARISON OF COMPUTATION TIMES REQUIRED BY THE NURBS-BASED AND 20-NODED SOLID FINITE ELEMENT MODELS .....	33

This page intentionally left blank

## 1.0 INTRODUCTION

---

One of the goals of the research project entitled “Investigation of LCM through Single Product Models (SPM)” is to automate the process of generation of structural finite element models from geometric information in SPM databases. NURBS (Non-Uniform Rational B-Spline) are a standard technology employed in SPM/CAD systems to represent geometry of surfaces such as ship hull forms. A numerical method for solving problems governed by partial differential equations, such as stress analyses of solids and structures, was developed recently based on NURBS representation of structural components. Although this method has many features in common with the traditional finite element method, and some features in common with the meshless methods, it is more geometrically based and has potential to directly utilize NURBS-based geometry definition in SPM/CAD systems to perform engineering analyses. It thus offers a number of advantages over the traditional finite element methods for integration with SPM databases.

The initial phase of this work involved implementation and verification of this NURBS-based computational method in the VAST finite element program [1]. The implementation was achieved as a new element type, IEC82, where the input data included the polynomial orders, the knots vectors, the control points and the weights. The verification problems included linear static, natural frequency, linear buckling and nonlinear analyses involving both geometric and material nonlinearities. The numerical results demonstrated a number of advantages of this NURBS-based formulation from the computational point of view. However, the test problems considered previously in [1] only involved relatively simple geometries, such as cylindrical and spherical shells, and no stiffeners were considered [1]. In addition, the convergence study performed in the previous phase was incomplete because the reference solutions employed in the study were obtained using a relatively coarse mesh of 20-noded solid elements. For linear buckling analyses, significant discrepancies were noticed between buckling loads and mode shapes predicted by the NURBS-based and conventional solid elements. Finally, the efficiency of the NURBS-based method needs to be tested using a more complex ‘ship-like’ structural component and a post-processing capability is still to be developed for viewing model meshes and results.

The present phase of the project is to address these remaining issues from the previous phase of work, as outlined above. The following work items are to be performed:

1. Conduct a more complete convergence study of the NURBS-based finite element for linear static analysis and compare the convergence properties of the NURBS-based solid element with that of the traditional 20-noded solid element.
2. Investigate the accuracy issue of the NURBS-based solid element for linearized buckling analysis, through a detailed review of the mathematical formulation and computer code and additional numerical verifications.
3. Compare the computational efficiency of the traditional 20-noded solid and NURBS-based solid elements for practical engineering problems. This task includes generation of 20-noded and NURBS-based solid element models of simplified ship structural

components and detailed comparison of efficiency and accuracy of these models for linear static analysis.

4. Develop a post-processing capability to automatically generate dummy 20-noded solid element models in knots space and to map and display displacement and stress results on to the dummy solid element models, so these 20-noded solid element models, along with the displacements and stress results, can be readily displayed with the existing graphics capability provided in Trident FEA.

All these tasks have been accomplished successfully and the details will be presented in the following chapters in this report. The conclusions from this study are given in Chapter 6.

## **2.0 DEVELOPMENT OF A POST-PROCESSING CAPABILITY FOR THE NURBS-BASED SOLID ELEMENT**

---

The development and verification of the post-processing capability is presented in this chapter. Although this is the last work item in the Statement of Work, we decided to discuss it first in the report because this capability is extensively utilized in the study of other work items in this contract.

As presented in the previous contract report [1], the “element” in the NURBS-based solid element formulation is defined by knot intervals in the knots space. However, differing from the traditional finite elements, the approximation space for the NURBS-based element is non-local, which means that the displacement field in an element depends not only on degrees-of-freedom of nodes inside the domain of the element, but also on nodes outside the element. This feature is in common with the meshless or mesh-free method.

In the post-processing capability, each “element” in a NURBS-based solid element model is subdivided into a uniform mesh of 20-noded solid elements. This subdivision is first performed in the knot space and then the actual coordinates of the nodes of the 20-noded solid elements are calculated from the coordinates of the control nodes in the three-dimensional physical space using NURBS functions. The numbers of elements in all three directions, used for this subdivision, are user controlled parameters, so a finer subdivision can always be conveniently achieved when higher resolutions are desired for result presentation.

When all “elements” in the NURBS-based model are subdivided, node equivalencing is required to eliminate the redundant nodes at the element interfaces. This operation is time-consuming, but is required to ensure stress contour plots can be displayed properly. At the present time, the post-processing capability is able to process results from both linear and nonlinear analyses, including displacements, stresses and plastic strains. Among them, the displacements are calculated using the displacements at the control points through the NURBS functions, just like the nodal coordinates.

The calculation of stresses and strains in the dummy 20-noded solid element model is a little different. In VAST, element stresses in the 20-noded solid element are stored at the 8 corner nodes and this format of stress storage is also adopted in the NURBS-based solid element. During post-processing, the stress variation in a NURBS-based element is approximated by a tri-linear function from which the stresses at corners of each element in the dummy 20-noded element model can be computed. In nonlinear analyses involving plastic deformations, stresses and plastic strains are calculated at the integration points. In this situation, the eight outmost integration points are used for the tri-linear interpolation. Alternative interpolation procedures, including higher-order interpolation functions and piecewise linear interpolations, have been explored. However, the results are not satisfactory.

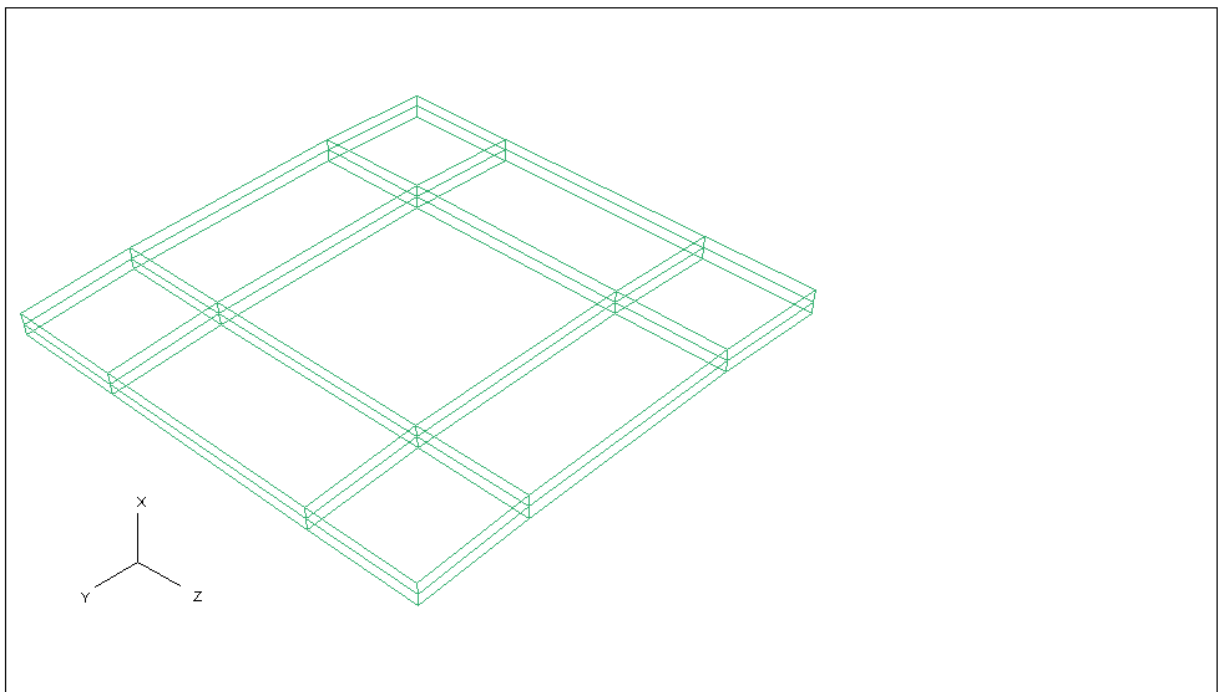
The post-processing program is developed in FORTRAN in a style consistent with the VAST code. As a result, it can be conveniently incorporated into VAST when it is required in the future.

In order to demonstrate and verify this post-processing capability, two example problems considered in the previous phase [1] were utilized. These test problems involved geometrically nonlinear analysis of a hinged cylindrical shell and combined geometrically and materially nonlinear analysis of a hinged spherical shell, both subjected to concentrated loads.

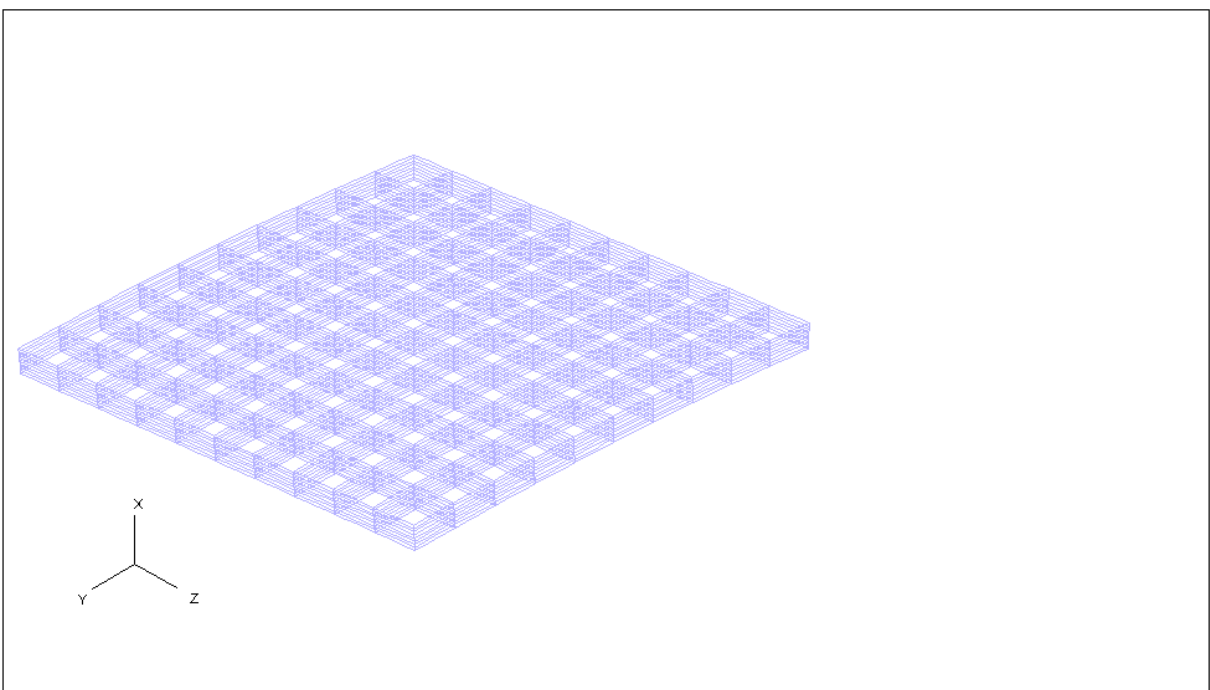
The control net of the NURBS-based solid element model, the dummy 20-noded solid element models created by the post-processor and the deformation contour plot generated by Trident using the dummy 20-noded solid element model are shown in Figures 2.1-2.3 and 2.4-2.6, respectively for these two test cases. These figures indicate that the nodal displacements of the dummy 20-noded solid element model are computed correctly. It is also interesting to note that although the outer edge of the control net for the second test problem is non-smooth, the 20-noded solid element model always represent the exact geometry of the structure. Actually, this is one of the advantages for using NURBS representation to describe a problem geometry.

For verification of the post-processing capability for stresses and strains, we utilized the second test case mentioned above, which involved a combined geometrically and materially nonlinear analysis of a hinged spherical shell subjected to a centre point load. As shown in Figure 2.4, in the NURBS-based solid element model, one quarter of the spherical shell was represented by three NURBS-patches. Each of these patches contains a  $4 \times 4 \times 1$  mesh of elements defined on knots intervals and during the post-processing, each element is subdivided into a  $4 \times 4 \times 10$  mesh of 20-noded solid. The reason for using ten elements in the thickness direction is to adequately capture through-the-thickness variations of stresses and plastic strains using Trident FEA, where the stress and strain are displayed based on values at element centroid. The generated stress and plastic strain contour plots for a typical load step are given in Figures 2.7 and 2.8, in which through-the-thickness stress and strain gradients are observed.

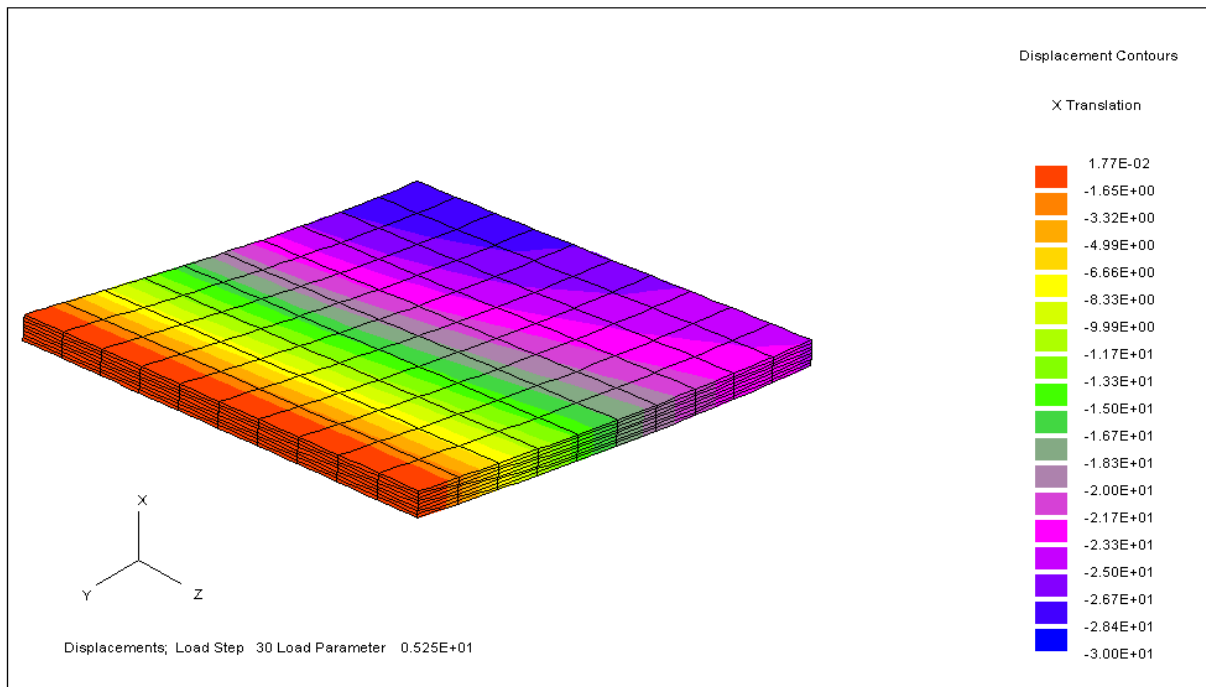




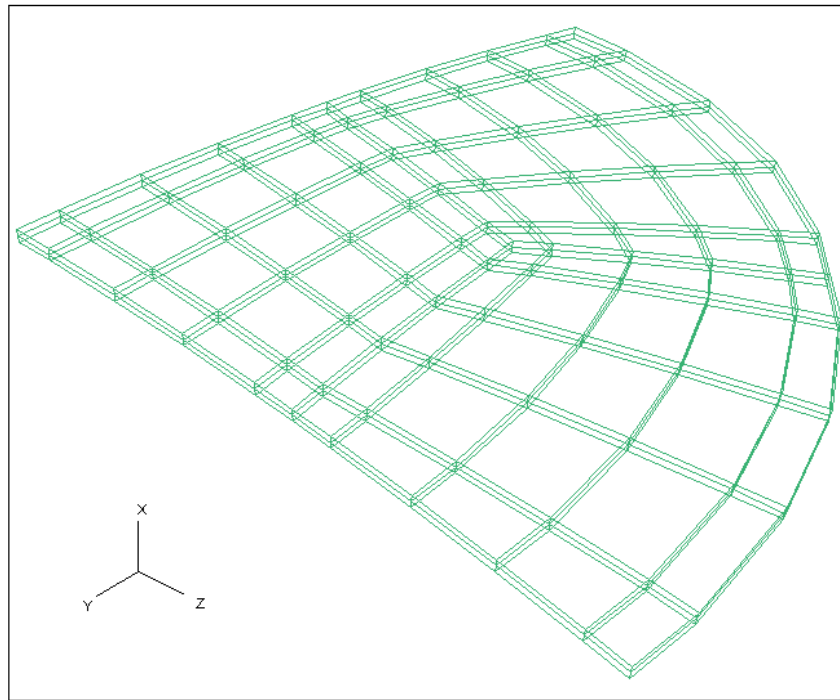
**Figure 2.1: Control Net of NURBS-based Solid Element Model of a Cylindrical Shell**



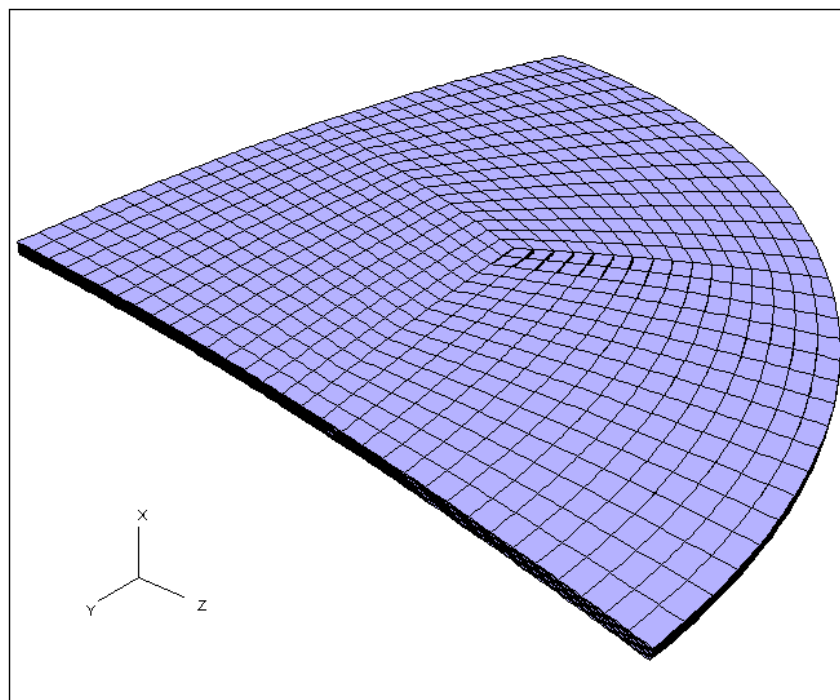
**Figure 2.2: A 20-Noded Solid Element Model of the Cylindrical Shell Generated by the Post-Processor**



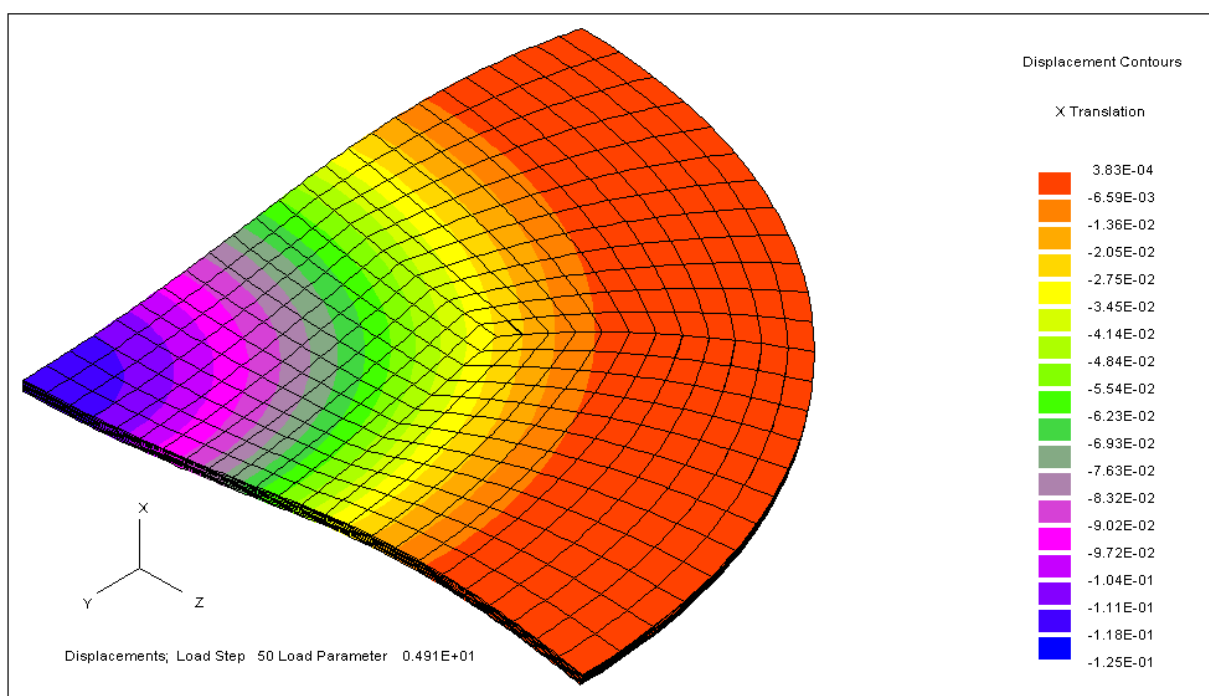
**Figure 2.3: Displacement Contour Generated Using the Post-Processing Capability for Geometrically Nonlinear Analysis of a Hinged Cylindrical Shell**



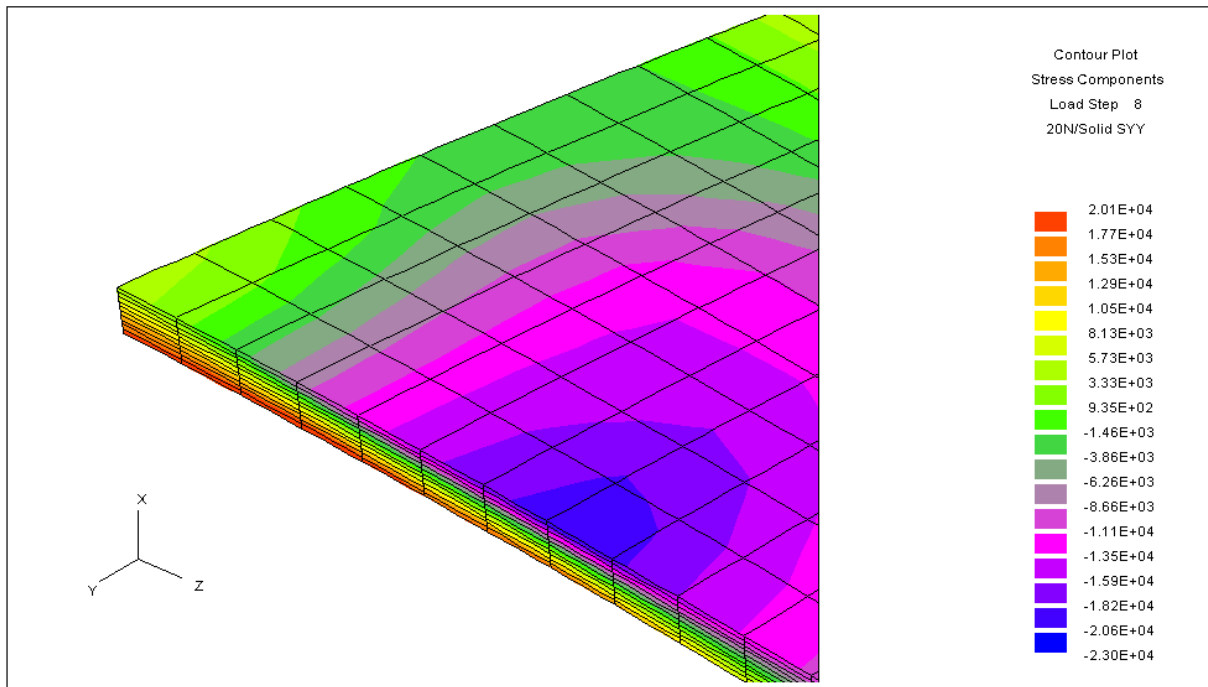
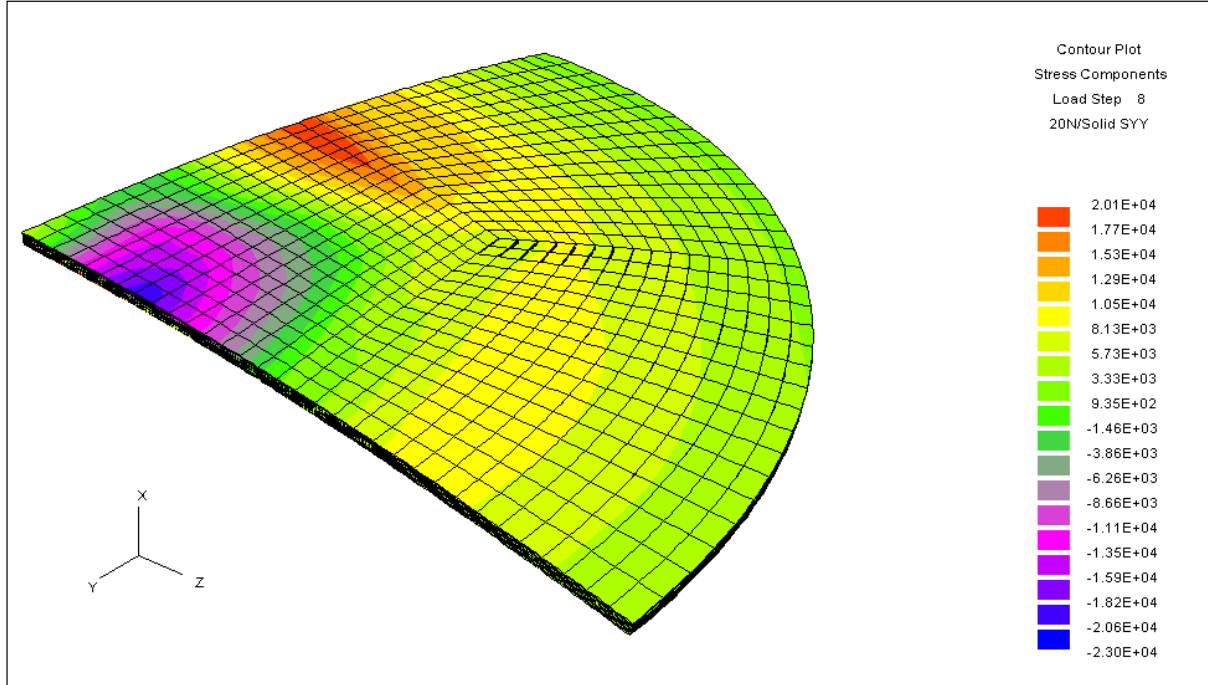
**Figure 2.4: Control Net of NURBS-based Solid Element Model of a Spherical Shell**



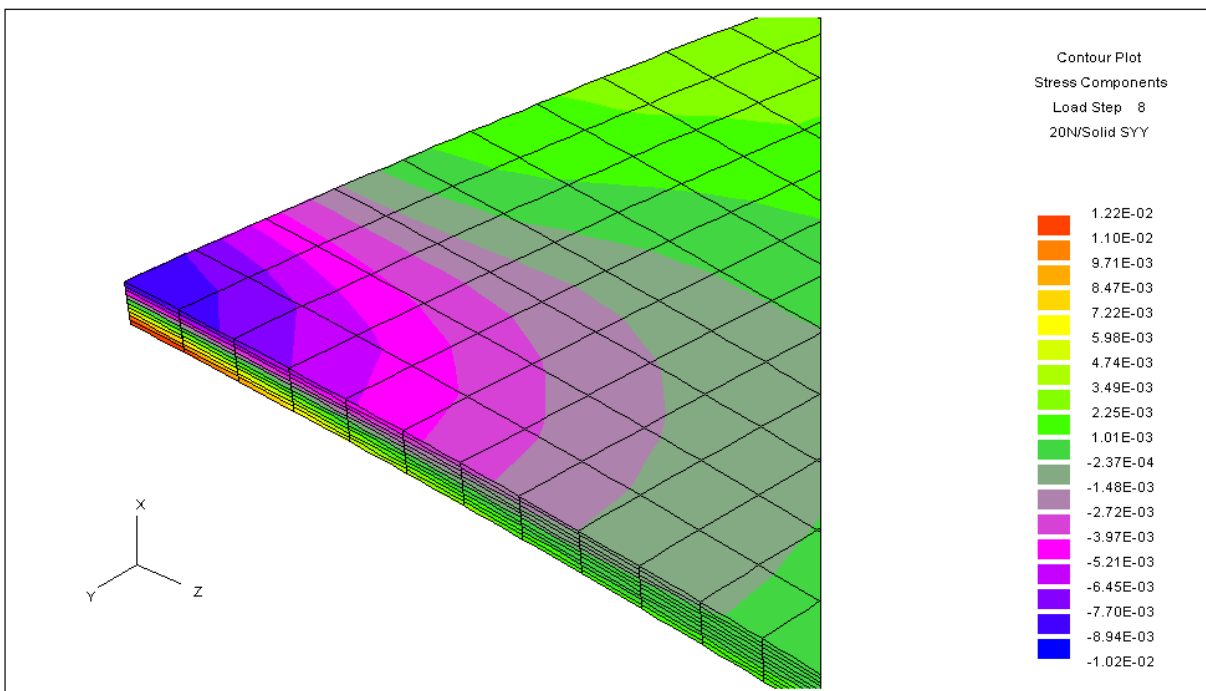
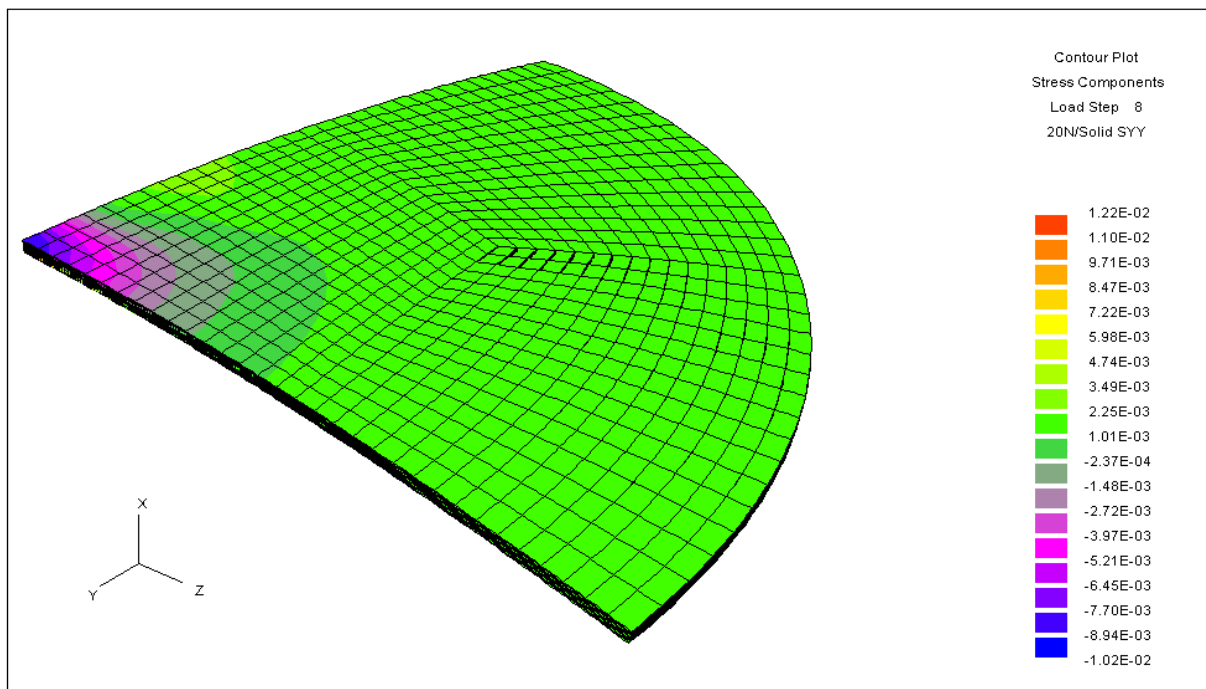
**Figure 2.5: A 20-Noded Solid Element Model of the Spherical Shell Generated by the Post-Processor**



**Figure 2.6: Displacement Contour Generated Using the Post-Processing Capability for Geometrically Nonlinear Analysis of a Hinged Spherical Shell**



**Figure 2.7: Stress Contour Plots Generated Using the Post-Processor at a Typical Load Step in Nonlinear Analysis of a Hinged Spherical Shell**



**Figure 2.8: Plastic Strain Contour Plots Generated Using the Post-Processor at a Typical Load Step in Nonlinear Analysis of a Hinged Spherical Shell**

### 3.0 CONVERGENCE STUDY OF THE NURBS-BASED SOLID ELEMENT FOR LINEAR STATIC ANALYSIS

---

In the previous phase of development of the NURBS-based solid element for VAST [1], the convergence characteristics of this element formulation were studied by considering linear elastic response of a hinged cylindrical shell subjected to a centre point load. Due to symmetry in the problem, only one quarter of the shell needed to be considered in the numerical solution. In the previous work [1], a variety of NURBS-based solid element models were constructed using different polynomial orders and different numbers of knot intervals. The displacement and stress results obtained by the NURBS-based solid element were compared against solutions of 20-noded solid element based on the conventional isoparametric formulation. However, at that time, only one relatively coarse mesh of the 20-noded solid element was considered, so the convergence property of the NURBS-based element was not fully verified.

In the present work, the previous convergence study was extended by refining the reference solutions using additional meshes of the conventional 20-noded solid element. These 20-noded solid element meshes could be divided into three groups based on their general layout. The meshes in the first group could be represented as  $N \times N \times 2$  and were used to evaluate the effect of mesh refinement in the in-plane directions. The second group contained meshes defined as  $4 \times 4 \times N$  and were used to reveal the effect of mesh refinement through-the-thickness direction. The final group contained a number of highly refined meshes, which were utilized to provide references for comparison. Four typical meshes of the 20-noded solid element used in the convergence study are displayed in Figure 3.1 and the results are summarized in Table 3.1 for both full and reduced integrations.

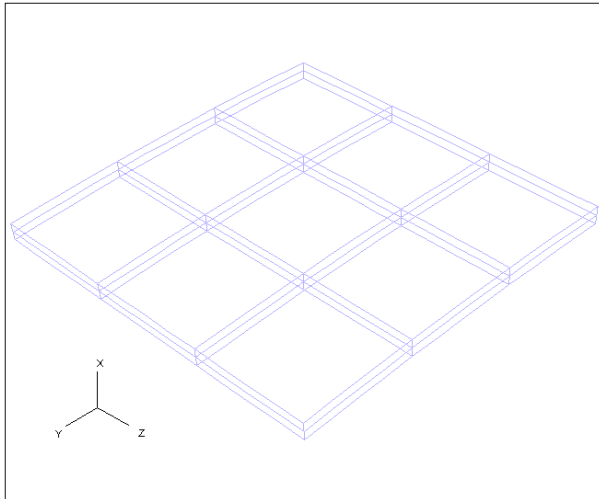
The newly obtained results for the 20-noded solid element were presented in Table 3.1 and also compared with the solutions of the NURBS-based solid element in Figures 3.2 and 3.3, where the results of the NURBS-based solid elements are taken from the previous report [1]. These comparisons indicate that for models of the same number of nodes, the NURBS-based solid element is far more accurate than the conventional 20-noded solid element. For example, the finest NURBS-based element model considered in the previous study contained  $13 \times 13 \times 2 = 338$  control points and predicted a centre deflection of 2.785. On the other hand, the centre deflection predicted by a  $5 \times 5 \times 2$  mesh (360 nodes) of the 20-noded solid element is only 2.487 for full integration and 2.597 when reduced integration was utilized. The prediction of the NURBS-based solid element was in a far better agreement with the reference solution of 2.848 obtained using a  $30 \times 30 \times 10$  mesh (40641 nodes) of the 20-noded solid element.

In addition, these results also indicated that for this particular test case, the accuracy of the 20-noded solid element is very sensitive to the order of numerical integration. Use of full integration results in “locking” of the element, so the convergence rate becomes extremely slow. The results of the NURBS-based element seem to be less sensitive to the numerical integration orders. All results for the NURBS-based element presented in Figures 3.2 and 3.3 were obtained using full integration.

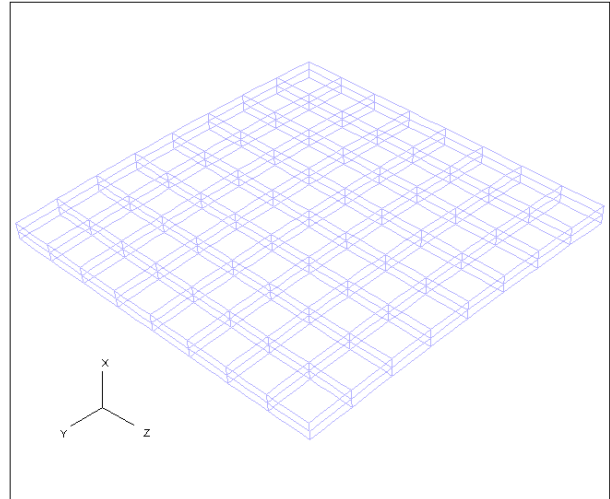
**Table 3.1: Results of Convergence Study Using 20-Noded Solid Element - Centre Deflection of a Hinged Cylindrical Shell Subjected to a Point Load**

Mesh	No. of Nodes	Full Integration	Reduced Integration
<b>Solid Element Mesh N×N×2</b>			
3×3×2	152	2.314	2.606
4×4×2	245	2.430	2.596
5×5×2	360	2.487	2.597
6×6×2	497	2.520	2.602
7×7×2	656	2.541	2.608
8×8×2	837	2.556	2.614
9×9×2	1040	2.568	2.620
<b>Solid Element Mesh 4×4×N</b>			
4×4×2	245	2.430	2.565
4×4×4	425	2.434	2.639
4×4×6	605	2.435	2.699
4×4×8	785	2.435	2.765
4×4×10	96	2.435	2.836
<b>Refined Solid Element Meshes</b>			
9×9×8	3320	2.577	2.697
15×15×10	10656	2.628	2.752
30×30×10	40641	2.717	2.848

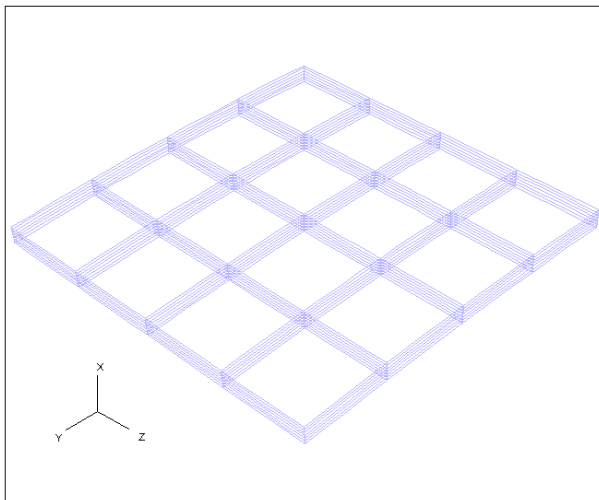




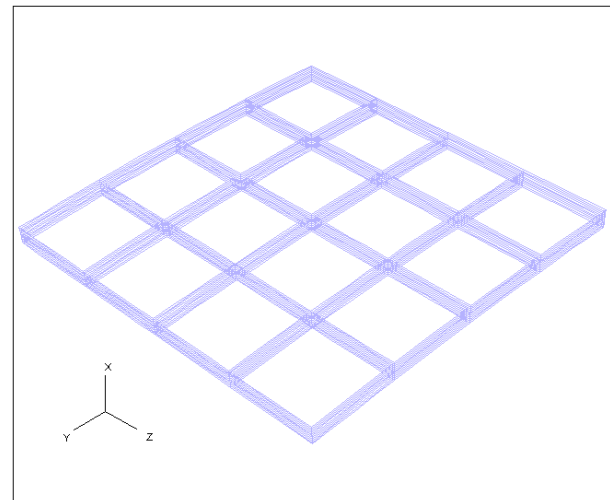
**(a) 3×3×2**



**(b) 8×8×2**

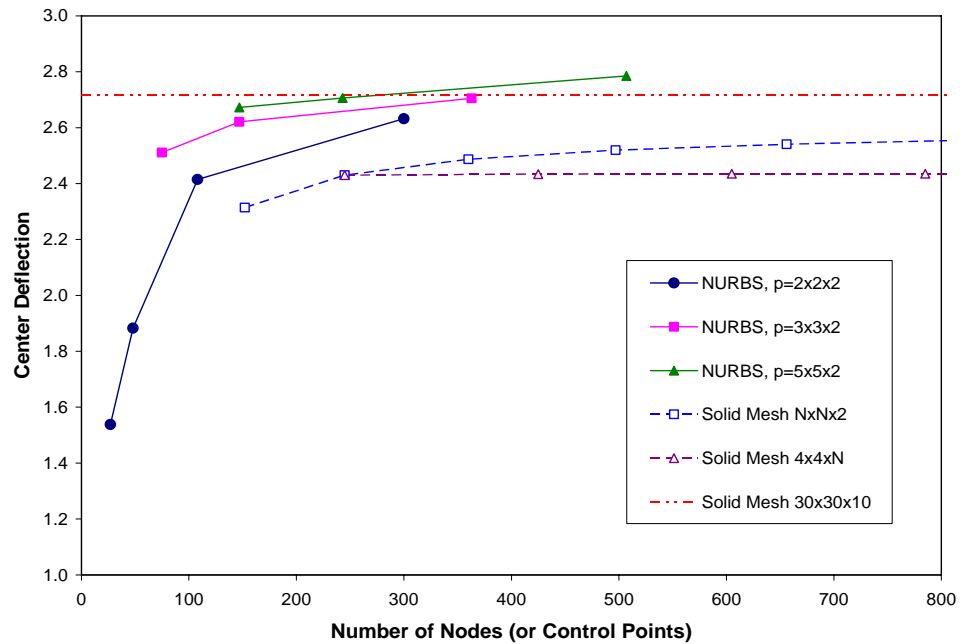


**(c) 4×4×6**

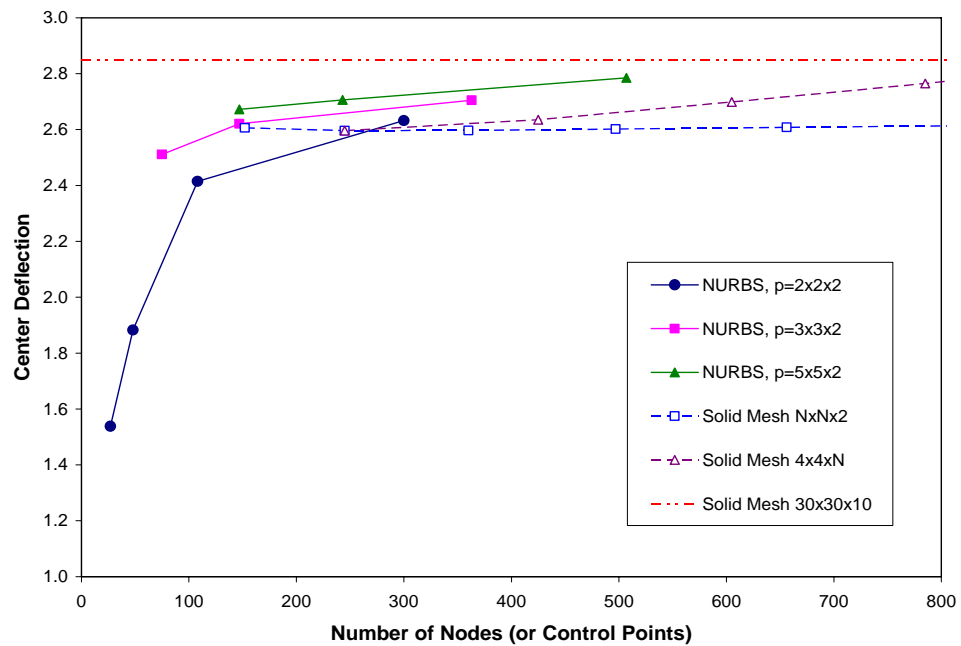


**(d) 4×4×10**

**Figure 3.1: Typical 20-Noded Solid Element Meshes Utilized in Convergence Study**



**Figure 3.2: Comparison of Centre Deflections Predicted by NURBS-based Solid Element and 20-Noded Solid Element with Full Numerical Integration**



**Figure 3.3: Comparison of Centre Deflections Predicted by NURBS-based Solid Element and 20-Noded Solid Element with Reduced Numerical Integration**

#### 4.0 AN INVESTIGATION OF ACCURACY OF THE NURBS-BASED SOLID ELEMENT FOR BUCKLING ANALYSIS

---

In the previous phase of the development of NURBS-based computational method for VAST [1], a test case involving linear buckling analyses of a hinged cylindrical shell subjected to a centre point load was considered. The problem geometry and loading condition are shown in Figure 4.1, where hinged boundary conditions are applied along the straight edges and the curved edges are stress free. Three NURBS-based models, which all contain the same number of elements but different orders of polynomial functions, were utilized in the verification. The control net for one of the NURBS-based solid element models is depicted in Figure 4.2, on which the boundary conditions and external load are also shown. During the earlier analyses, the minimum orders of numerical integration, which produced non-singular stiffness matrices, were employed and the predicted buckling load factors for the first 5 modes were compared to the results of a 20-noded solid element model. The figure showing this comparison is copied from the previous report [1] and presented in Figure 4.3. These results indicated that the second and fifth modes obtained by the NURBS-based solid elements are not converging with the increase of the polynomial order.

In order to better understand the problem, we first compared the buckling mode shapes for Modes 2 and 5 predicted by the various models. The buckling mode shapes obtained using the 20-noded solid element and the NURBS-based solid element models with the third- and fifth-order polynomials are given in Figures 4.4, 4.5 and 4.6, respectively. These results indicated that the mode shapes predicted by the 20-noded solid and the NURBS-based element with 3<sup>rd</sup>-order basis functions are consistent, but the buckling modes predicted by the NURBS-based model with higher order polynomials are very different.

To identify potential errors introduced during the implementation of the NURBS-based solid element, the source code of VAST was then checked carefully, with special attention paid to the sections for generating geometric stiffness matrix and calculating stresses at the numerical integration points from the corner values. However, no bugs were discovered.

In order to gain more insight of this problem, it is desirable to consider a simpler test example having known analytical solutions. To this end, a simply-supported flat square plate subjected to uniform bi-axial compression, as shown in Figure 4.7, was utilized. For this problem, the critical compressive stresses at the edges are expressed as [2]

$$\sigma_{cr} = \frac{\pi^2 D}{a^2 h} (m^2 + n^2)$$

where  $a$  and  $h$  indicate the edge length and thickness of the square plate, and  $D$  is the flexural rigidity of plate defined as  $D = Eh^3/12(1-\nu^2)$ . Parameters  $m$  and  $n$  determine the number of half-waves in the X and Y directions.

Four different models of the NURBS-based solid element were created to represent one quarter of the square plate. These models contained the same numbers of control points, but

different polynomial orders. As a result, the number of “elements”, which were defined by knot intervals, varied from model to model because the size of knot vectors was different in these models. The control net of a representative NURBS-based model is also given in Figure 4.7.

Numerical calculations were first performed using the NURBS-based model with the highest polynomial order ( $5 \times 5 \times 2$ ), but different orders of numerical integration. The predicted critical stresses for the first ten buckling modes by using different integration orders are compared in Table 4.1, where the corresponding analytical solutions are also shown. The numbers of half-waves,  $m$  and  $n$ , were determined by visually examining the buckling mode shapes using the post-processing capability presented in the preceding chapter of this report. The mode shapes for two representative buckling modes are shown in Figure 4.8, which correspond to  $m=n=3$  and  $m=n=5$ , respectively. The results presented in Table 4.1 and Figure 4.8 suggested that to achieve reliable and accuracy buckling loads, the orders of numerical integration should be at least identical to the polynomial orders used in the NURBS-based solid element model. These results also indicated that the numerically predicted critical stresses are consistently lower than the analytical solutions. This is because the analytical solutions are derived using the plate theory in which through-the-thickness deformations are neglected.

Based on the results presented in Table 4.1, the VAST solver was modified to set the default orders of numerical integration equal to the polynomial orders in each of the parametric directions. To confirm the robustness of this default integration rule, additional calculations were carried out using other NURBS-based models using the default integration orders. The numerical results are summarized in Table 4.2, where a stable convergence pattern is observed.

At this point, the buckling problem for the hinged cylindrical shell, depicted in Figure 4.1, was reanalyzed using the default numerical integration orders. The new results of buckling load factors for the first five modes are shown in Figure 4.9, where convergence is observed for all the modes. The buckling mode shapes obtained by the NURBS-based solid element model with 5<sup>th</sup>-order polynomials using the default numerical integration orders are shown in Figure 4.10. Comparing these buckling mode shapes with those shown in earlier figures, Figures 4.4-4.6, it is readily recognized that the errors observed in the previous study [1] were caused by the inadequacy of the numerical integrations.

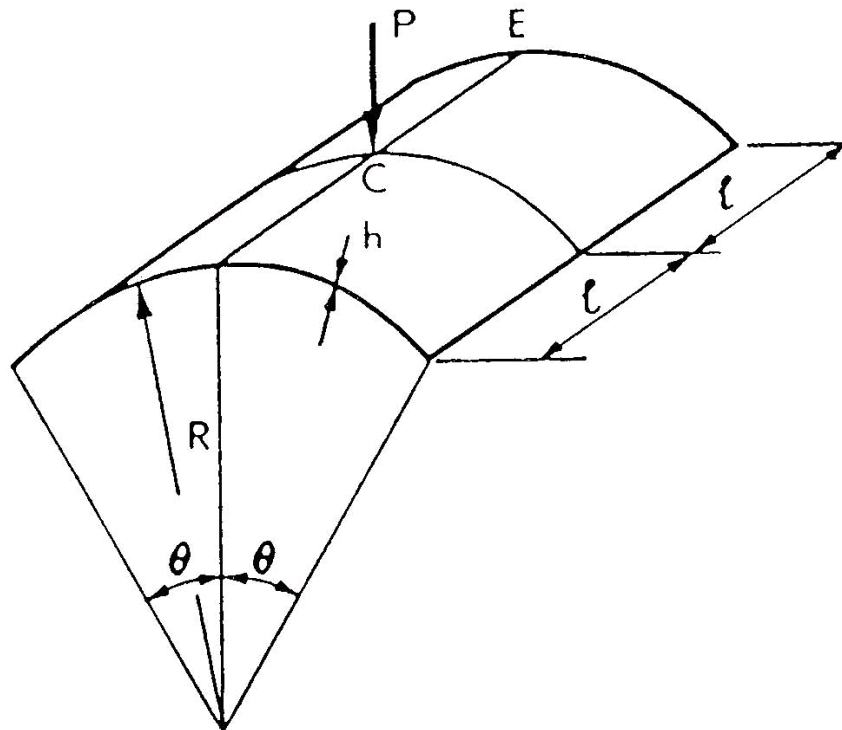
In order to see whether this inadequacy can be observed by examining the solutions of linear static analyses, we compared displacement and stress contours produced by the same NURBS-based model, but with different numerical integration orders in Figures 4.11-4.12. Surprisingly, these results are in very close agreement. These observations suggest that higher numerical integration orders are required to accurately integrate the geometric stiffness matrix than required for the material stiffness matrix.

**Table 4.1: Comparison of Critical Stresses for Buckling of a Bi-Axially Loaded Simply Supported Square Obtained by a NURBS-based Solid Element Model Using Different Orders of Numerical Integration**

Mode #	NURBS-based Finite Elements (MPa): Numerical Integration Order $n_u \times n_v \times n_s$				Analytical (MPa)		
	3×3×2	4×4×2	5×5×2	8×8×3	m	n	Critical $\sigma$
1	58.37	58.87	58.89	58.89	1	1	59.31
2	293.47	294.41	294.45	294.45	3	1	296.56
3	293.49	294.42	294.45	294.46	1	3	296.56
4	520.94	525.70	525.86	525.88	3	3	533.81
5	759.97	761.11	761.18	761.21			
6	759.99	761.13	761.19	761.23			
7	979.93	987.14	987.39	987.45	5	3	1008.31
8	980.13	987.26	987.50	987.55	3	5	1008.31
9	1425.2	1438.70	1439.10	1439.30	5	5	1482.81
10	1454.2	1467.40	1468.80	1469.00			

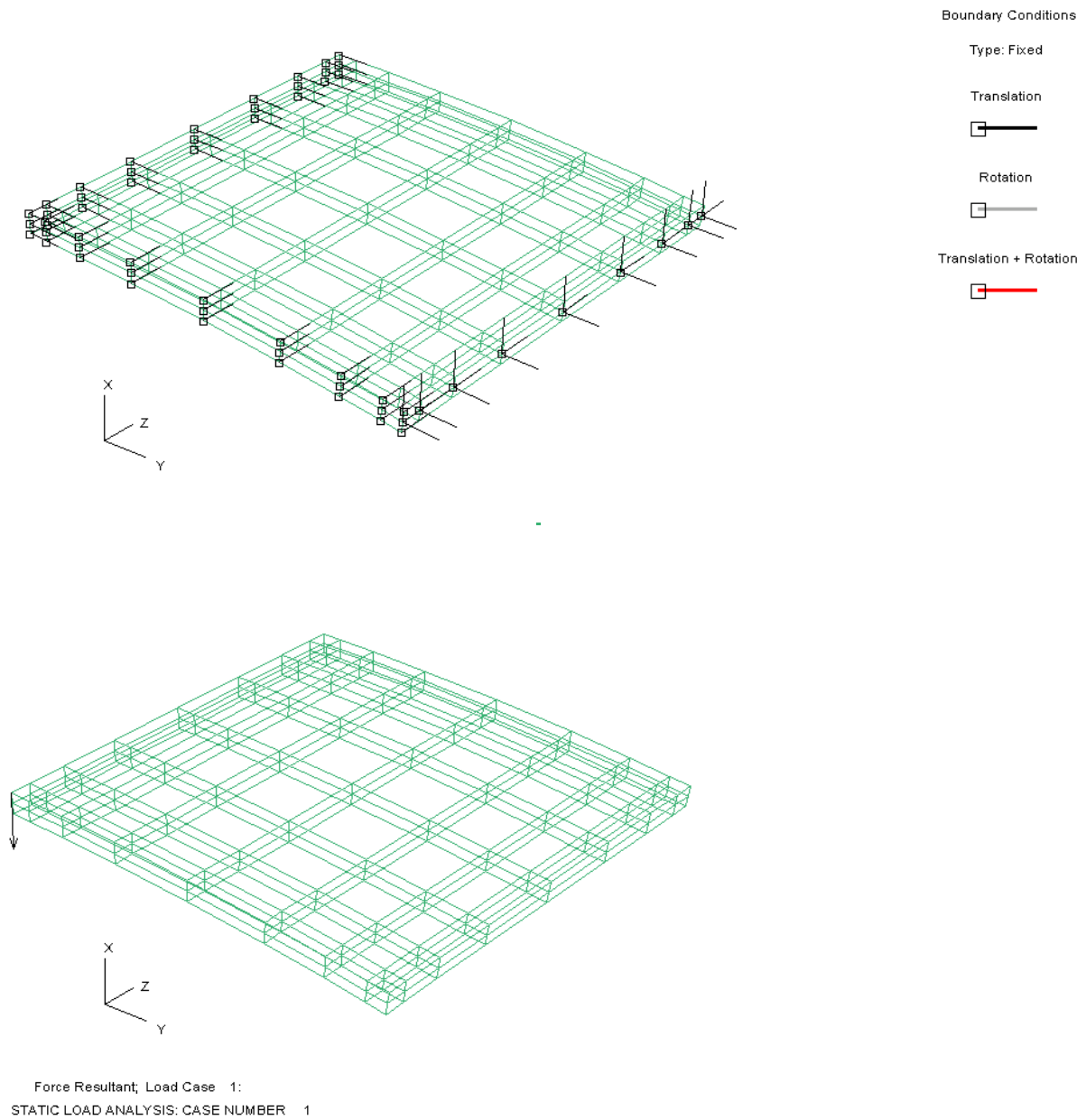
**Table 4.2: Comparison of Critical Stresses for Buckling of a Bi-Axially Loaded Simply Supported Square Obtained by Different NURBS-based Solid Element Models Using the Default Numerical Integration**

Mode #	NURBS-based Finite Elements (MPa): Polynomial Order $p_u \times p_v \times p_s$				Analytical (MPa)		
	2×2×2	3×3×2	4×4×2	5×5×2	m	n	Critical $\sigma$
1	59.00	58.96	58.91	58.89	1	1	59.31
2	296.24	294.77	294.50	294.45	3	1	296.56
3	296.25	294.78	294.51	294.45	1	3	296.56
4	528.72	526.69	526.10	525.86	3	3	533.81
5	797.99	768.73	762.31	761.18			
6	798.00	768.74	762.32	761.19			
7	1017.80	993.93	988.52	987.39	5	3	1008.31
8	1017.90	994.02	988.63	987.50	3	5	1008.31
9	1481.50	1448.20	1440.80	1439.10	5	5	1482.81
10	1738.00	1545.90	1487.00	1468.80			

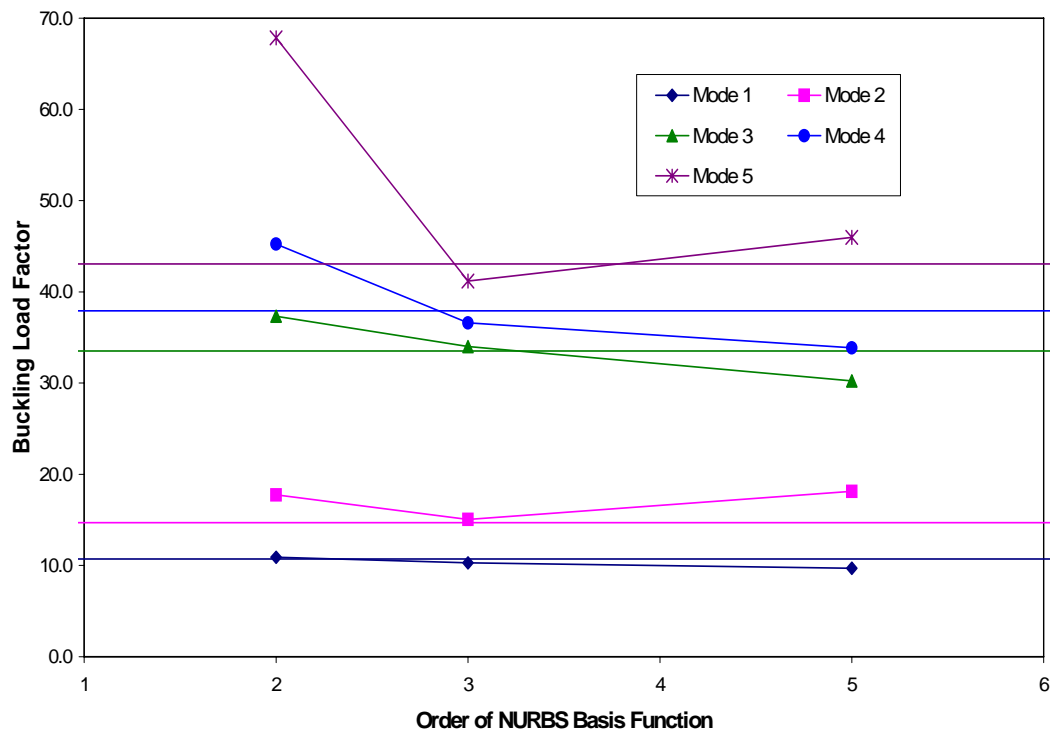


$$\begin{aligned}
 R &= 2540 \text{ mm} \\
 l &= 254 \text{ mm} \\
 h &= 12.7 \text{ mm}, 6.35 \text{ mm} \\
 \theta &= 0.1 \text{ rad.} \\
 E &= 3.10275 \text{ kN/mm}^2 \\
 \nu &= 0.3
 \end{aligned}$$

**Figure 4.1: Problem Definition for Buckling Analyses of a Hinged Cylindrical Shell Subjected to a Centre Point Load**

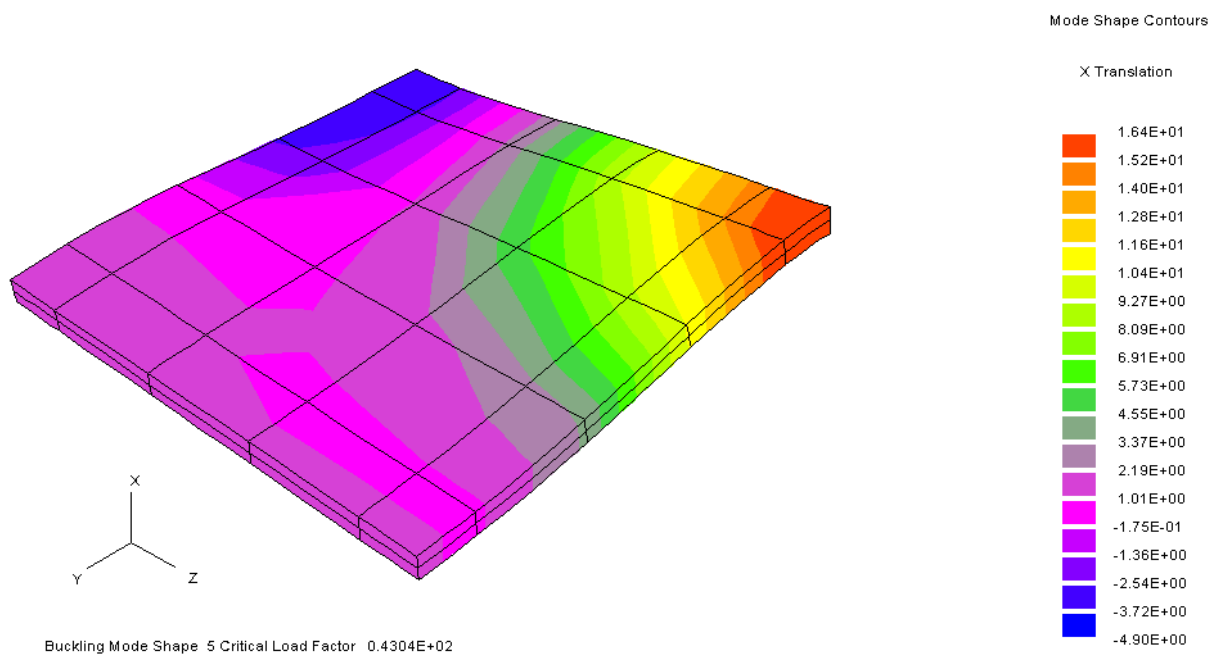
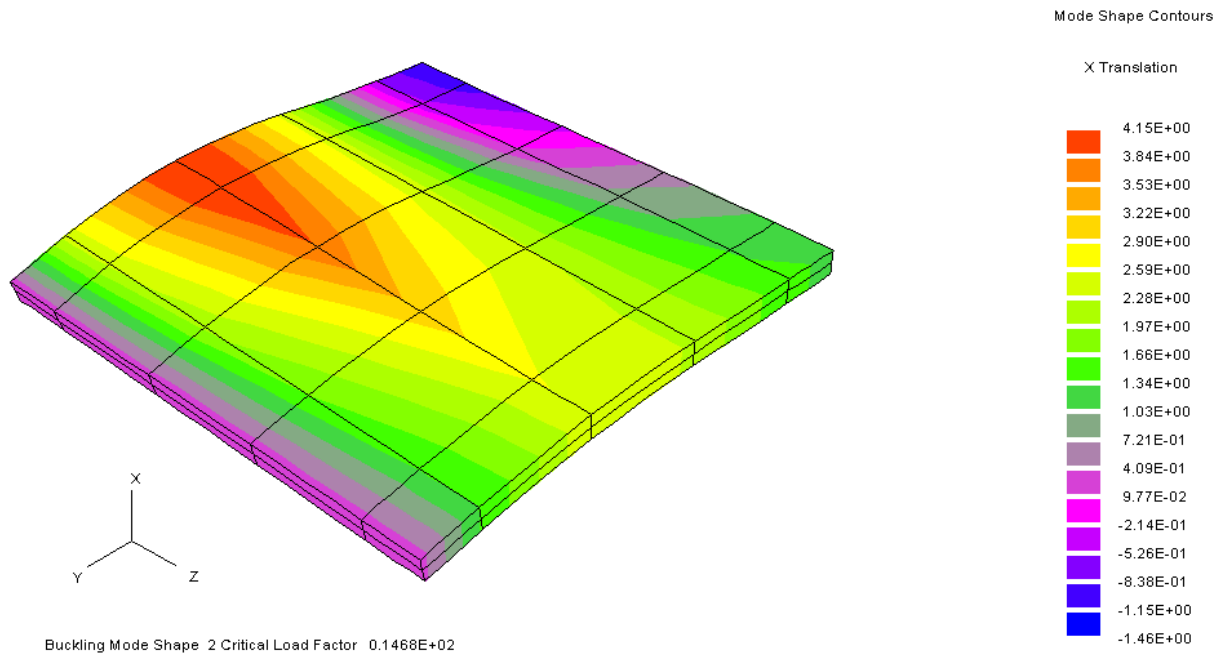


**Figure 4.2: Control Net of a NURBS-based Solid Element Model Representing One Quarter of the Cylindrical Shell with Boundary Conditions and External Load**

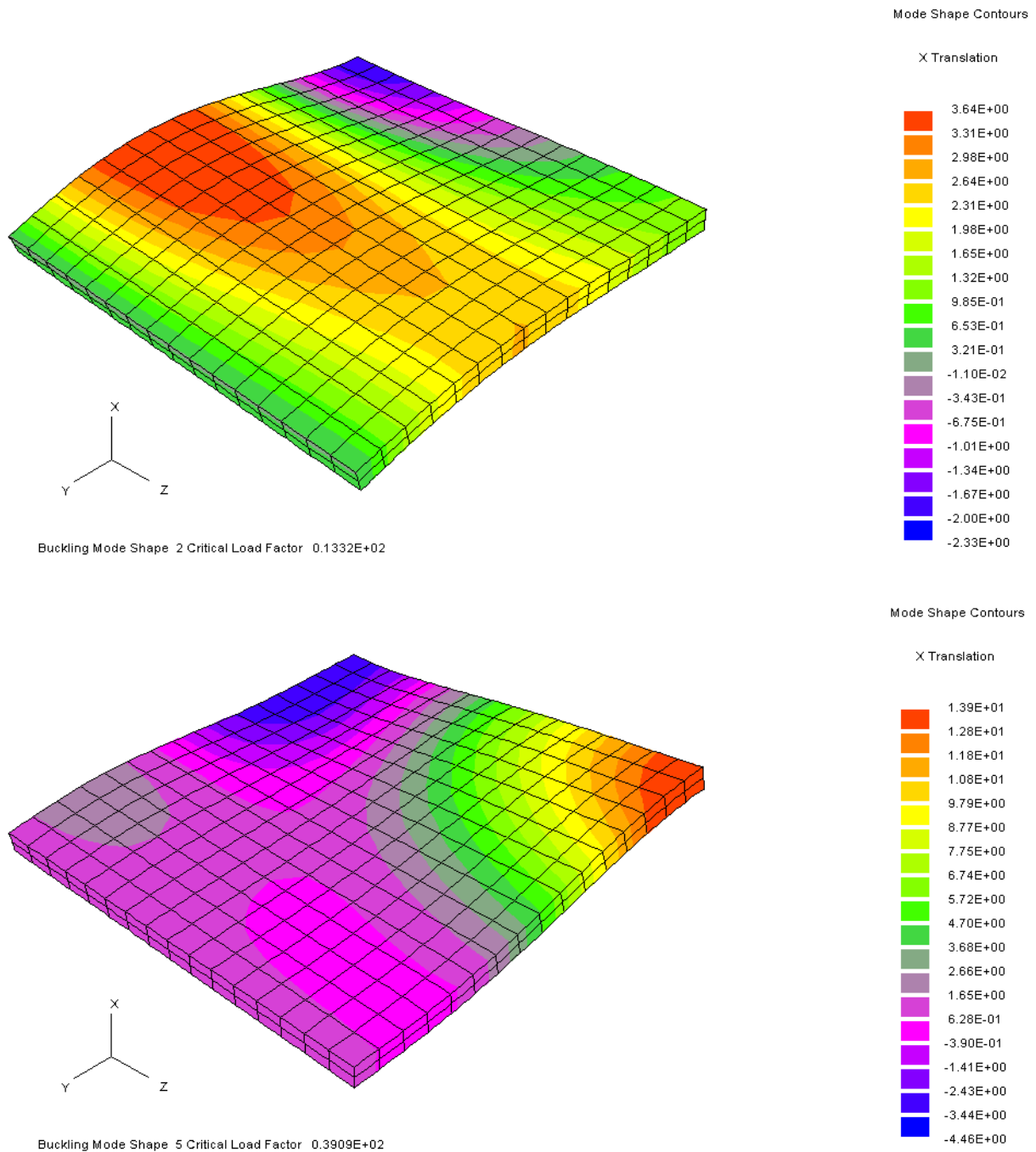


**Figure 4.3: Buckling Loads Obtained by NURBS-based Solid Elements of Various Orders and 20-Noded Solid Elements Presented in Previous Contract Reported [1]**

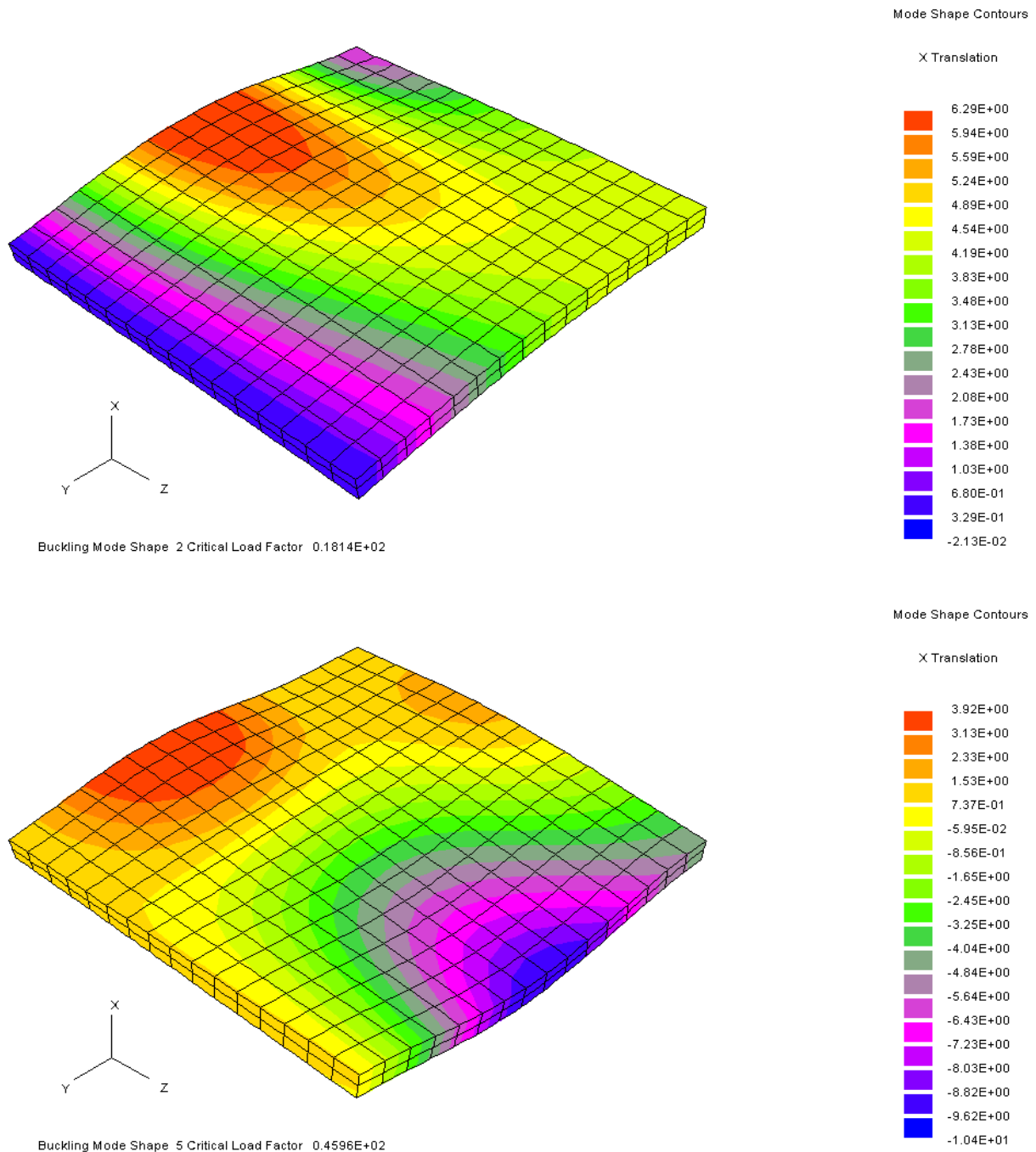




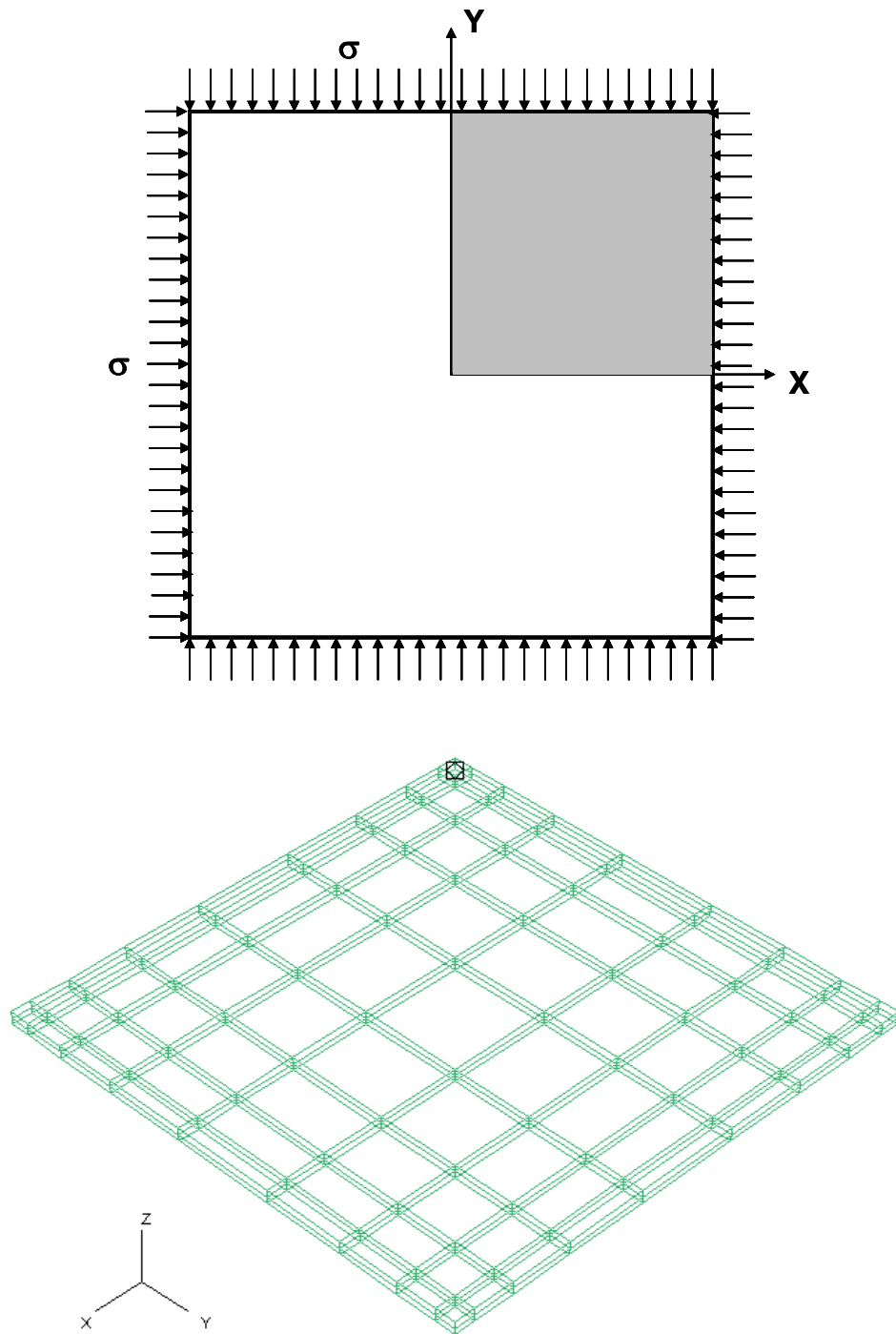
**Figure 4.4: Buckling Mode Shapes for Mode 2 and 5 Predicted by the 20-Noded Solid Elements**



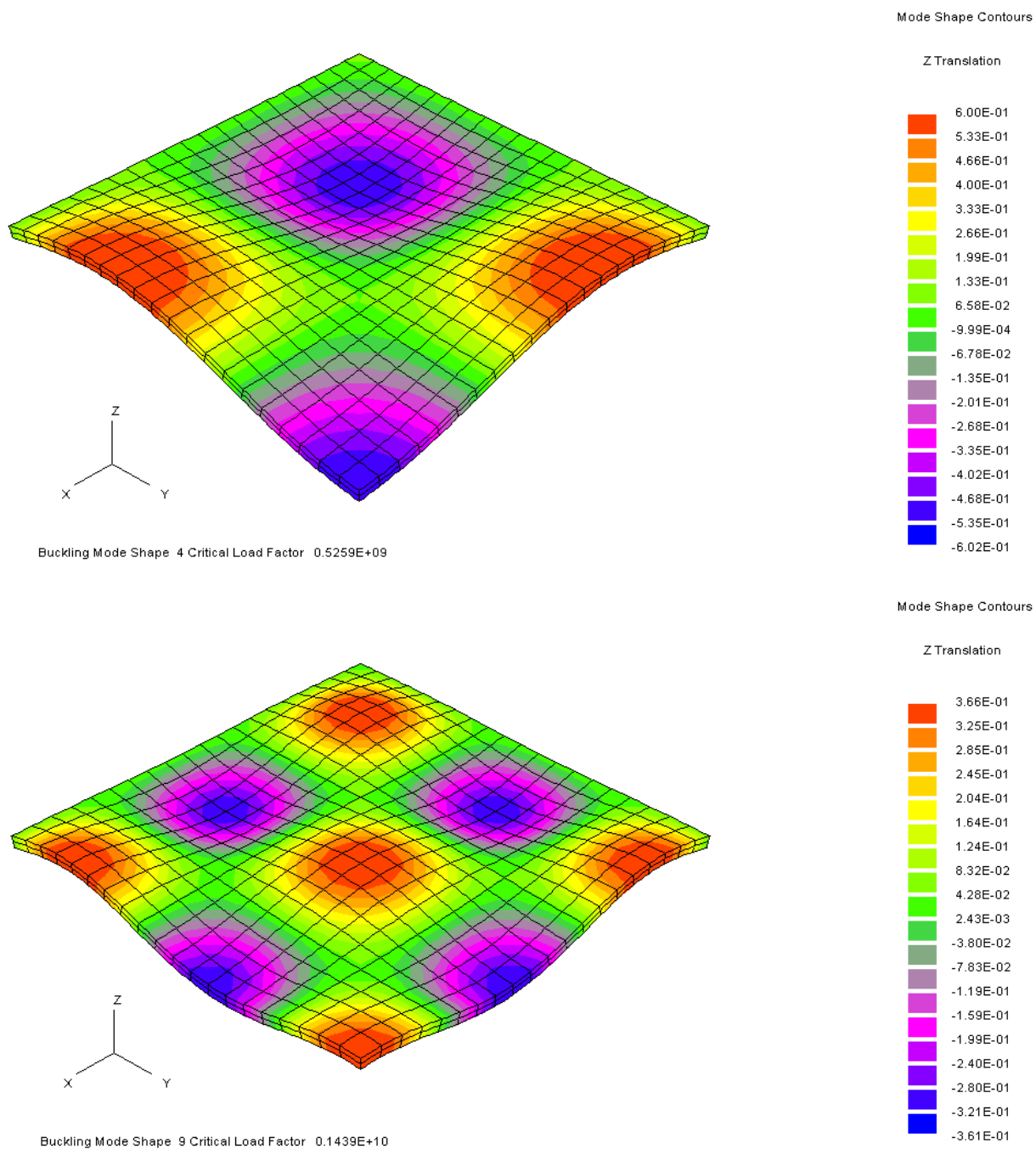
**Figure 4.5: Buckling Mode Shapes for Mode 2 and 5 Predicted by NURBS-based Solid Element Model of Order 3×3×2 Using 3×3×2 Numerical Integration**



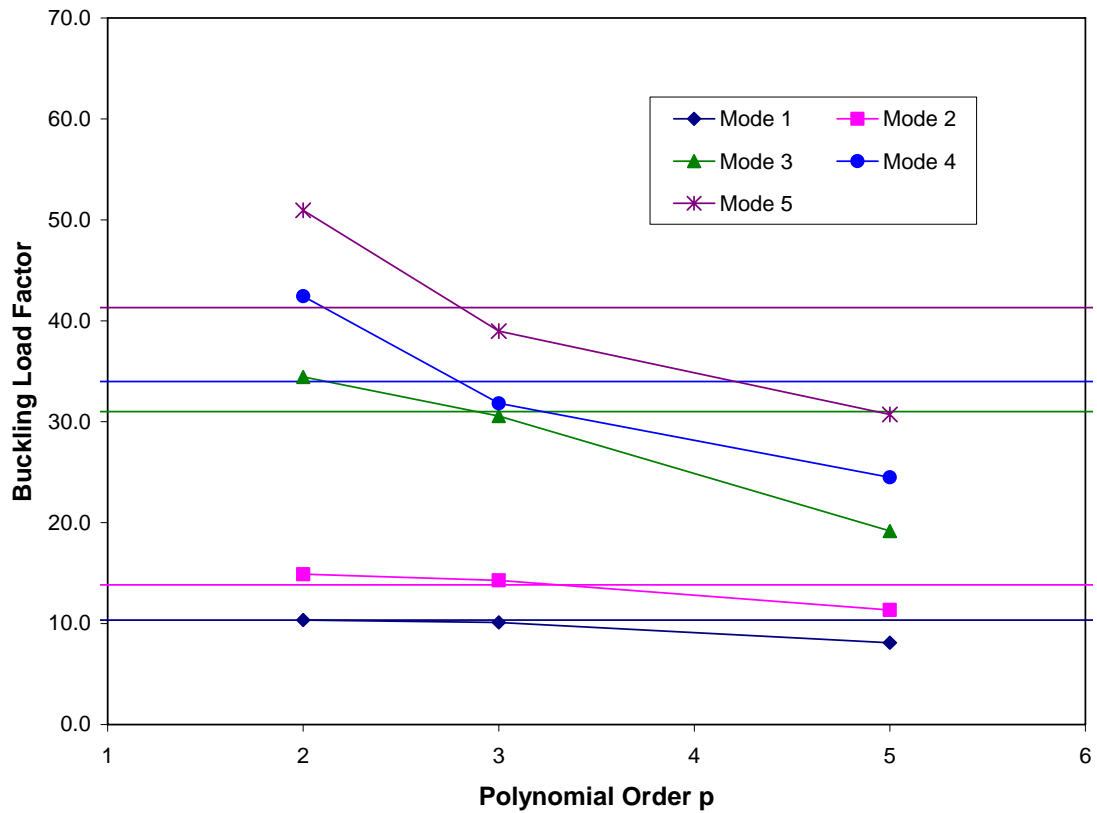
**Figure 4.6: Buckling Mode Shapes for Mode 2 and 5 Predicted by NURBS-based Solid Element Model of Order 5×5×2 Using 4×4×3 Numerical Integration**



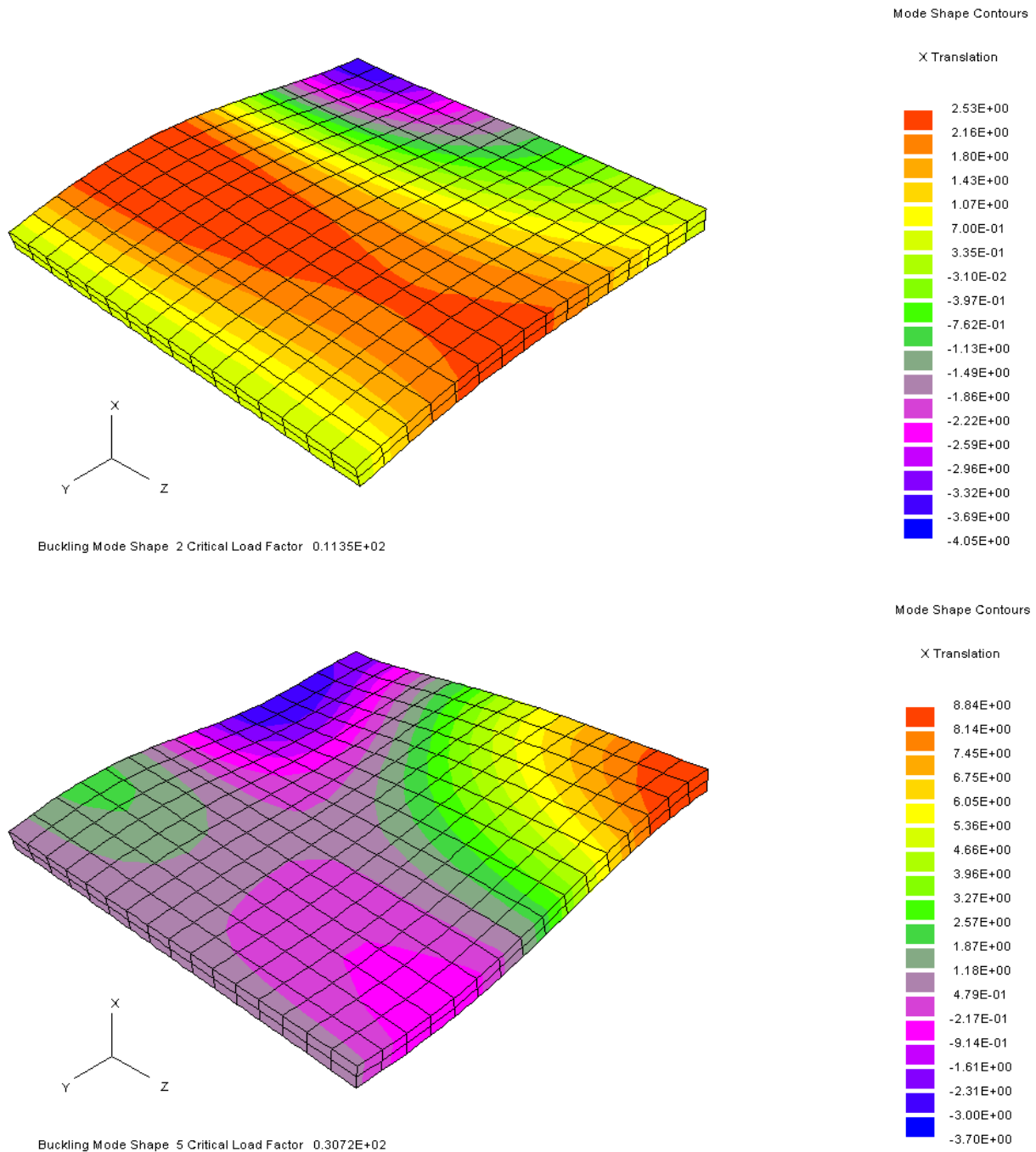
**Figure 4.7: Problem Definition for Buckling of a Square Plate Subjected to Biaxial Compression and the Control Net of a NURBS-based Solid Element Model of Order  $5 \times 5 \times 2$  Representing One Quarter of the Plate (Shaded Area)**



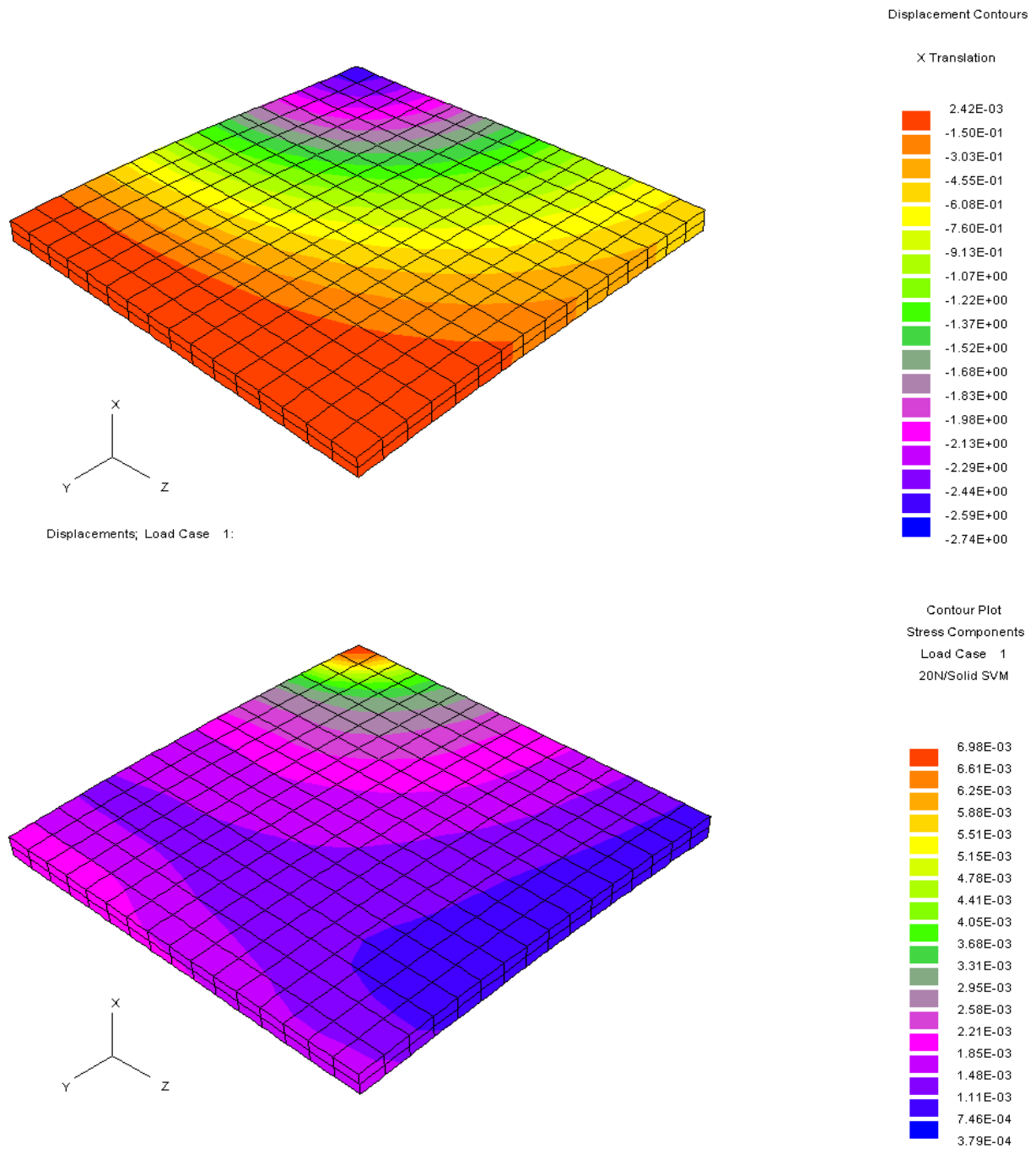
**Figure 4.8: Buckling Modes 4 and 9 of the Bi-axially Loaded Plate Predicted by NURBS-based Solid Element Model of Order  $5 \times 5 \times 2$  Using  $8 \times 8 \times 3$  Numerical Integration**



**Figure 4.9: Buckling Load Factors Obtained by NURBS-based Solid Elements of Various Polynomial Orders Using Proposed Default Orders of Numerical Integration**

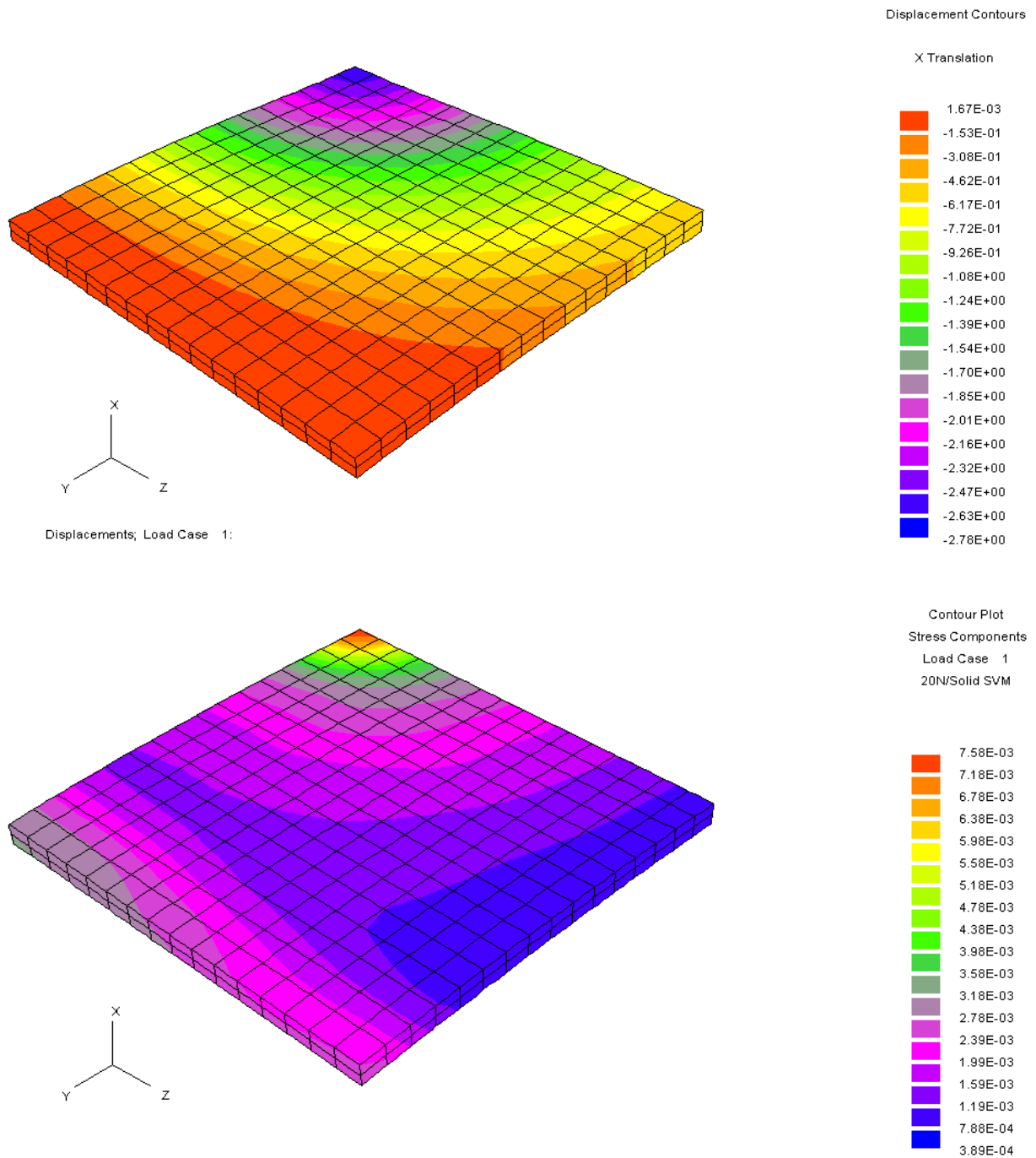


**Figure 4.10: Buckling Modes 2 and 5 of the Cylindrical Shell Predicted by NURBS-based Solid Element Model of Order 5×5×2 Using 5×5×2 Numerical Integration**



**Figure 4.11: Linear Static Solutions Obtained Using NURBS-Based Solid Element Model of Polynomial Order  $5 \times 5 \times 2$  Using  $4 \times 4 \times 3$  Numerical Integration**





**Figure 4.12: Linear Static Solutions Obtained Using NURBS-Based Solid Element Model of Polynomial Order  $5 \times 5 \times 2$  Using  $5 \times 5 \times 2$  Numerical Integration**

## 5.0 COMPARISON OF COMPUTATIONAL EFFICIENCY OF THE NURBS-BASED AND TRADITIONAL SOLID ELEMENTS FOR SHIP STRUCTURE ANALYSES

---

In order to benchmark the computational efficiency of the NURBS-based solid element against the traditional 20-noded isoparametric solid element for analysis of practical ship structures, a simplified ship hull component was considered. This simplified model included hull plating, a truncated transverse bulkhead and two stiffeners with Tee section, as shown in Figure 5.1. The original model was a surface model. In the present work, this surface model was first reverse-engineered utilizing Intelliship (Intergraph Corporation, [www.intergraph.com](http://www.intergraph.com)) to create a solid model.

The solid model was created with help of Rhino3D (McNeel Corporation, [www.rhino3d.com](http://www.rhino3d.com)). The procedures used to create a solid model based on the surface model include the following steps:

1. Offset each surface in the surface model by the distances determined by the structural object thickness to generate surfaces representing top, bottom and interior of the structural components.
2. Trim and extend the offset surfaces at the intersections of structural components.
3. Create new surfaces for the solid entities caused by the thicknesses of the intersecting components.
4. Sew all the surfaces to form solids.

After the solid model was completed, it was used to create an isogeometric NURBS model, which can be defined as:

- It must be a solid model with 6 surfaces.
- Two opposite surfaces must have the same NURBS definitions, e.g. top and bottom surfaces must have the same degree, number of control points and same knot vectors.
- If two solid patches are connected, then the two surfaces that touch each other must have identical NURBS definitions.

In order to create a NURBS model, which satisfies all the above requirements, one command for checking the surface parametric directions is implemented for Rhino3d. During the model generation, a new set of surface definitions was recreated for all the surfaces in the solid model so that all the requirements for NURBS model definitions were fulfilled. Then with help of the Rhino3d command, the orientation of the surface parametric directions was fixed. The final solid model generated using this approach is shown in Figure 5.2. This model includes NURBS representation of each individual solid patch involved in the finite element analysis. However, to obtain a NURBS-based solid element model useable for stress analysis, node equivalencing must still be performed to properly connect these solid patches. A special purpose FORTRAN program was developed to perform this task of model integration.

There were some issues encountered during the creation of the solid model. The most important ones are presented below:

- Rhino3d gives errors when trying to trim the hull with bulkhead surfaces (or even with a plane). This problem is avoided by increasing the thickness from 7mm to 10mm. This is a very serious problem because the CAD tool cannot handle the required geometry. One solution is to implement a new application to do the trimming using third party geometric kernel software, such as TSNLib (Spatial Corporation) or ACIS (SML Corp), export the model into standard file formats like IGES, and then import the model into the CAD tool. However, this is a time consuming procedure.
- When creating a NURBS representation of a solid entity, any of the commercial CAD tools provide user controls for the math definitions of the surfaces that define a solid patch. These math definitions of surfaces include polynomial orders, knot vectors and number and distribution of the control points. However, in a NURBS-based solid element model, not only must the math definitions of opposite surfaces in a given solid patch be identical, but the math definitions of the contiguous surfaces between two connecting solid patches must also be identical as well. In order to resolve this difficulty, separate tools were developed in the present work. However, the tools that were developed for this work are neither general-purpose nor based on any commercial CAD tools. To create models for the NURBS-based solid element efficiently, an automatic tool based on commercial CAD programs like Rhino3d should be developed.

To perform linear static analysis, clamped boundary conditions were applied to the two sides and the bottom edge of the model. A concentrated force were applied close to the intersection of the bulkhead and one of the stiffeners on the outside of the ship hull pointing inwards in the direction normal to the surface as depicted in Figure 5.3. Although these selections of the boundary and loading conditions did not correspond to a practical loading case on ship structures, they provided sufficient constraints and loads to the numerical model to permit comparison of the local deformations and stresses at the junctions of different structural components predicted by various finite element formulations. The undeformed control net and the deformed configuration of the stiffened panel are shown in Figure 5.4, where a summary of the displacements at one of the loaded nodes is also shown. This plot of deformed shape was generated using the post-processing capability for the NURBS-based finite element developed in this contract.

In order to compare the computational efficiency of the NURBS-based finite element and traditional 20-noded isoparametric solid element, two meshes of the 20-noded solid elements were constructed for the same stiffened plate component with the aid of the post-processing capability for NURBS-based element. In one of the solid element meshes, each element in the NURBS-based solid element model, which is defined by knot intervals in the parametric space, was converted into one 20-noded solid element. This mesh is shown in Figure 5.5. In the second solid element mesh, each element in the NURBS-based model was subdivided into eight 20-noded solid elements as shown in Figure 5.6. Linear static analyses were performed using both solid element models. The deformed configurations predicted by the solid elements are given in Figures 5.5 and 5.6 along with summaries of nodal displacements. A comparison of the displacements indicated that the accuracy of the solutions from the NURBS-based solid element model is comparable to those obtained using the finer mesh of the 20-noded solid

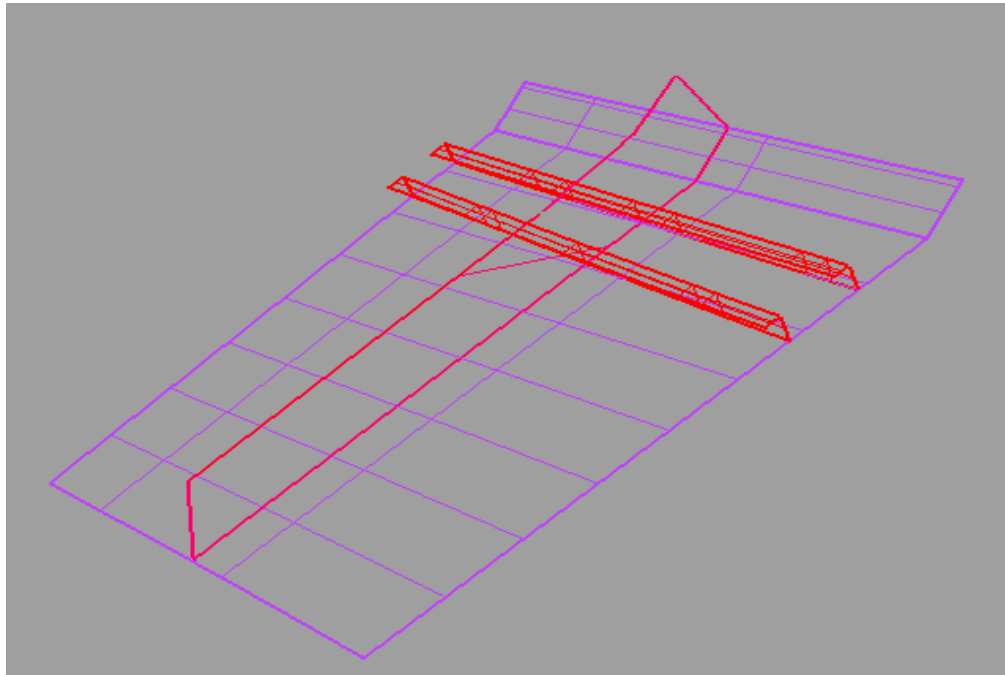
element. However, as indicated in Table 5.1, the computation time required by the NURBS-based model is even less than that required by the coarse mesh of the 20-noded solid element.

In order to further evaluate the relative performance of the NURBS-based and 20-noded solid elements, the von Mises stress contours generated by the various models were compared. The stress distribution predicted by the NURBS-based elements is shown in Figure 5.7, where both the entire model and the local details in the vicinity of the applied loads are displayed. The solutions generated by the two 20-noded solid element meshes for the same local area are presented in Figure 5.8. A comparison of these results revealed that the stress solution from the NURBS-based solid element model has a significantly higher resolution than the results of the coarse solid element model.

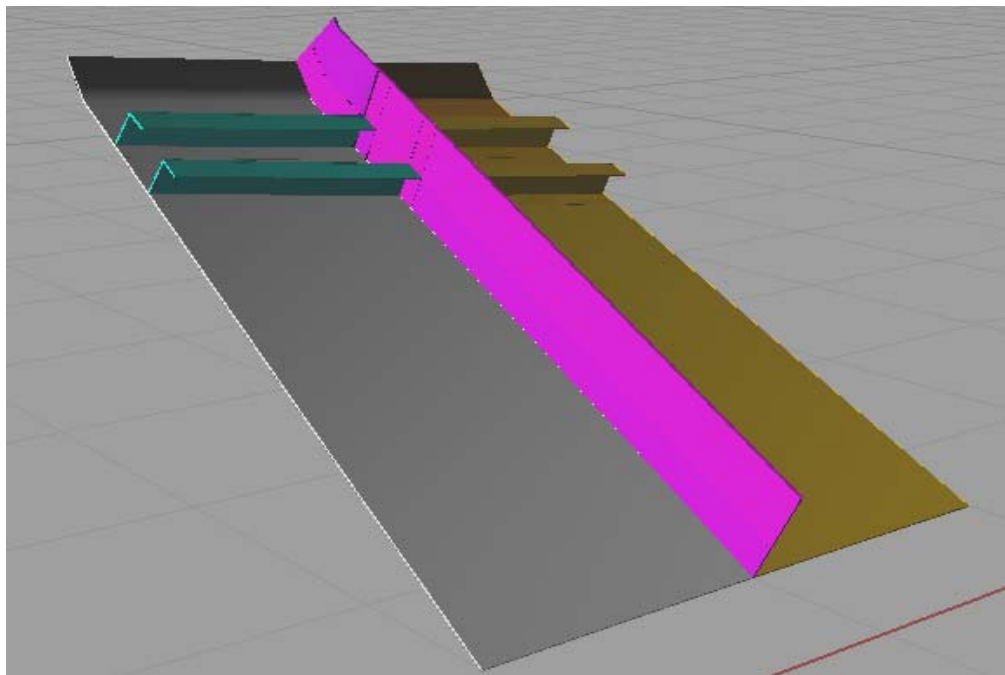
These results clearly demonstrated the potential of the NURBS-based finite element for ship structure analyses.

**Table 5.1: Comparison of Computation Times Required by the NURBS-based and 20-Noded Solid Finite Element Models**

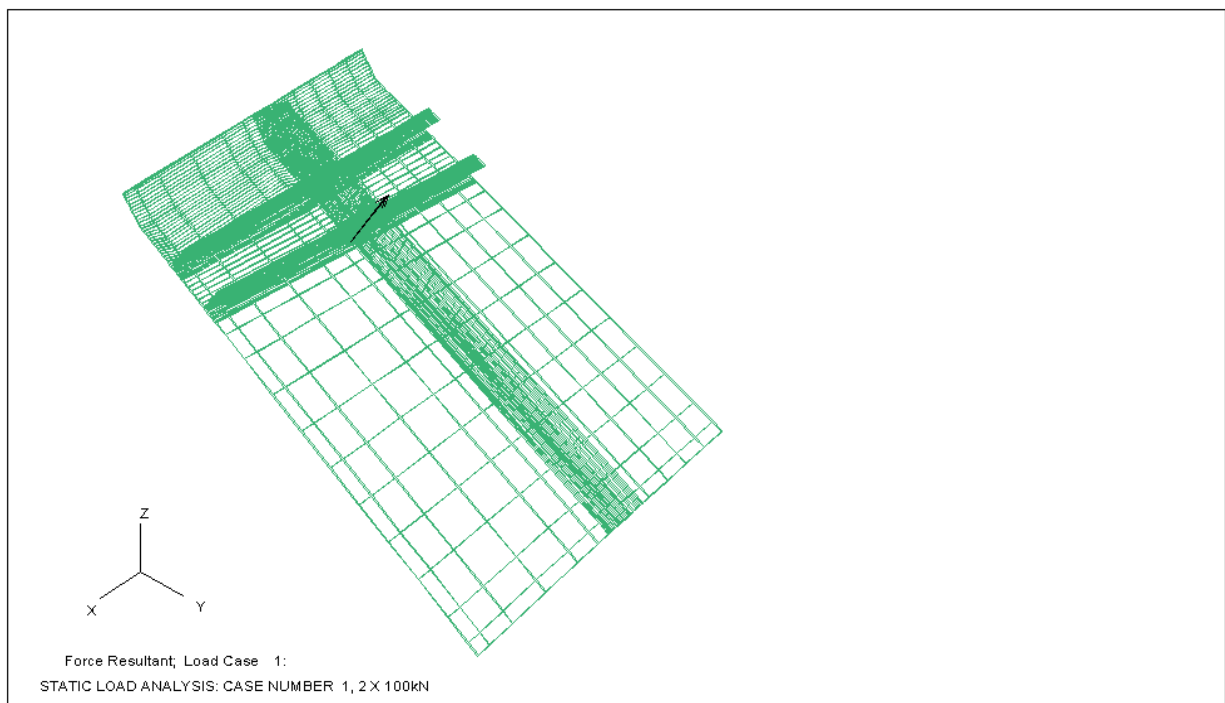
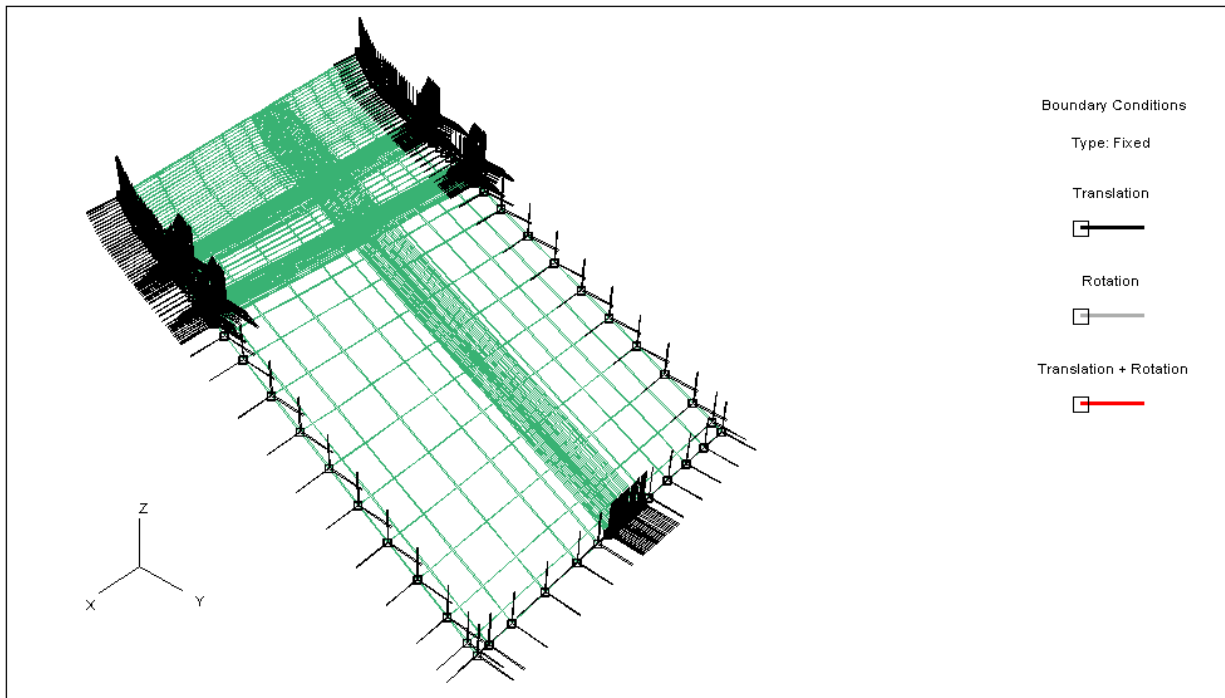
Model	No. of Nodes	Elapsed Time By Module (sec)			Total Time (sec)	Ratio
		ELEMS1	DECOM	STRESS		
NURBS-based	5532	14.297	12.218	1.594	28.594	1:1
Solid (coarse)	12071	7.953	30.500	0.656	39.453	1.38:1
Solid (fine)	74513	41.594	774.593	41.859	968.140	33.66:1



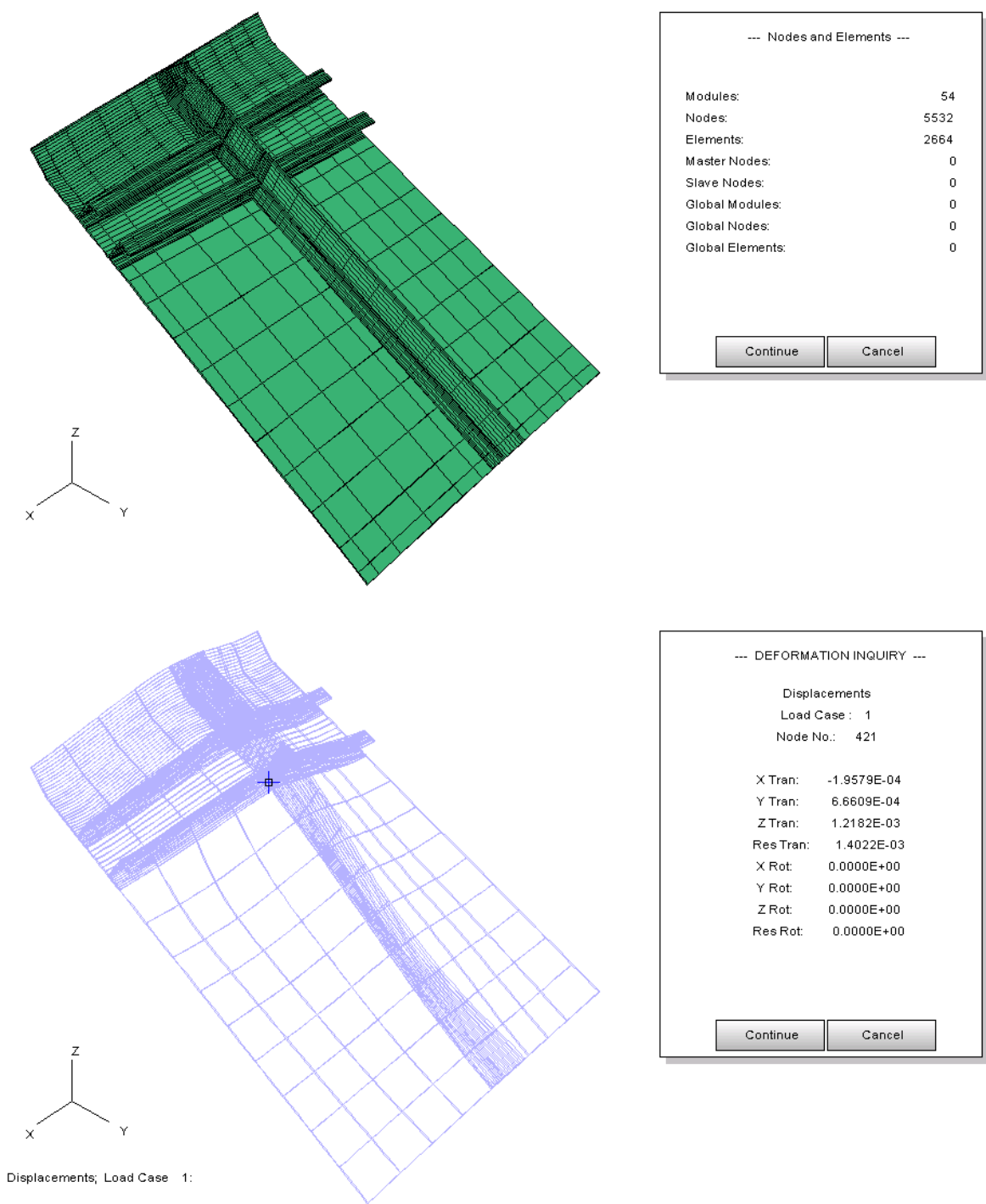
**Figure 5.1: Surface Model of a Stiffened Plate Structural Component**



**Figure 5.2: Isogeometric NURBS Model of the Stiffened Plate Structural Component**

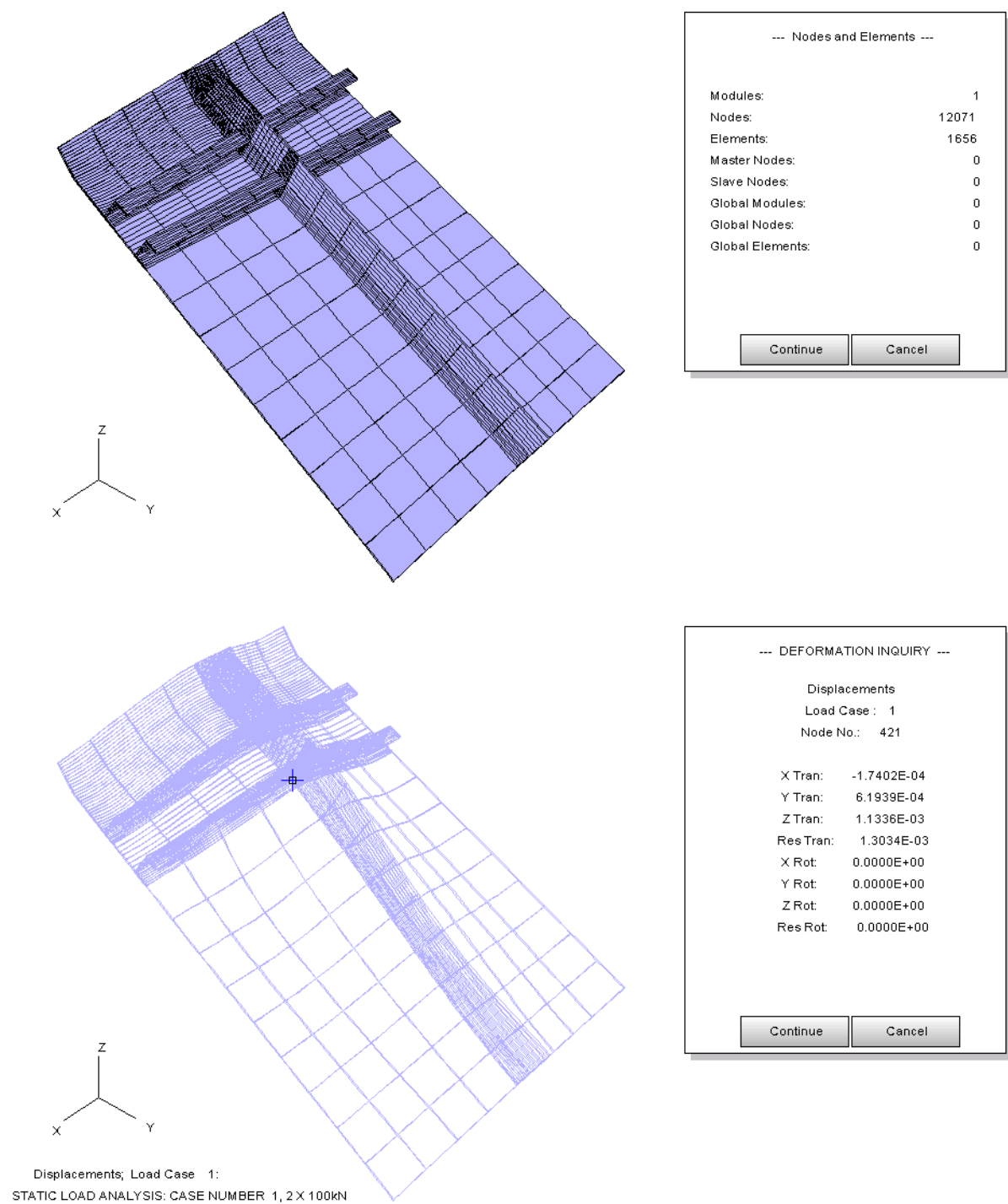


**Figure 5.3: Control Net of the NURBS-based Solid Element Model of the Stiffened Plate Structural Component Shown with Boundary Conditions and External Load**

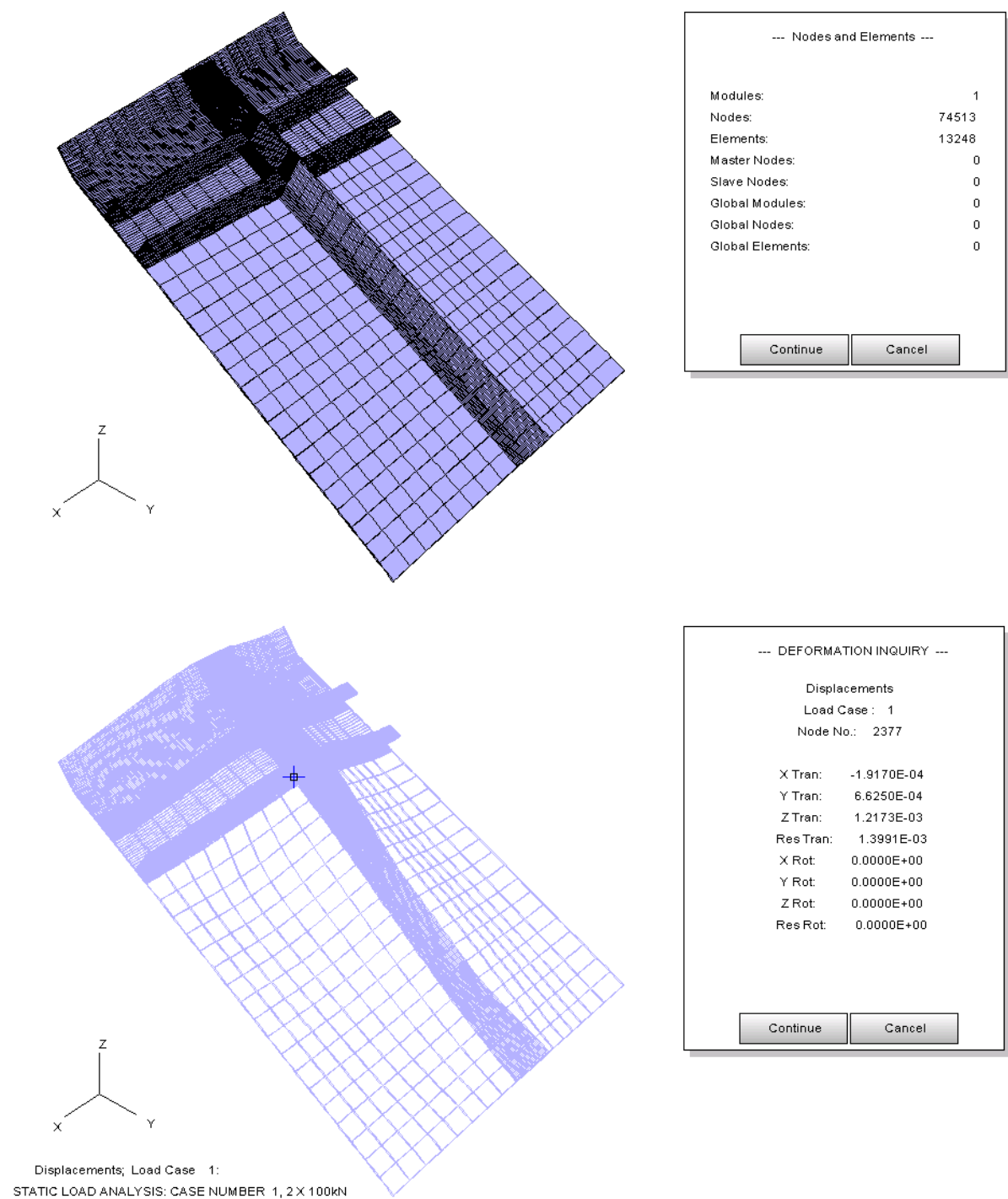


**Figure 5.4: Undeformed Control Net and Deformed Configuration of a NURBS-based Solid Element Model of the Stiffened Plate Structural Component**

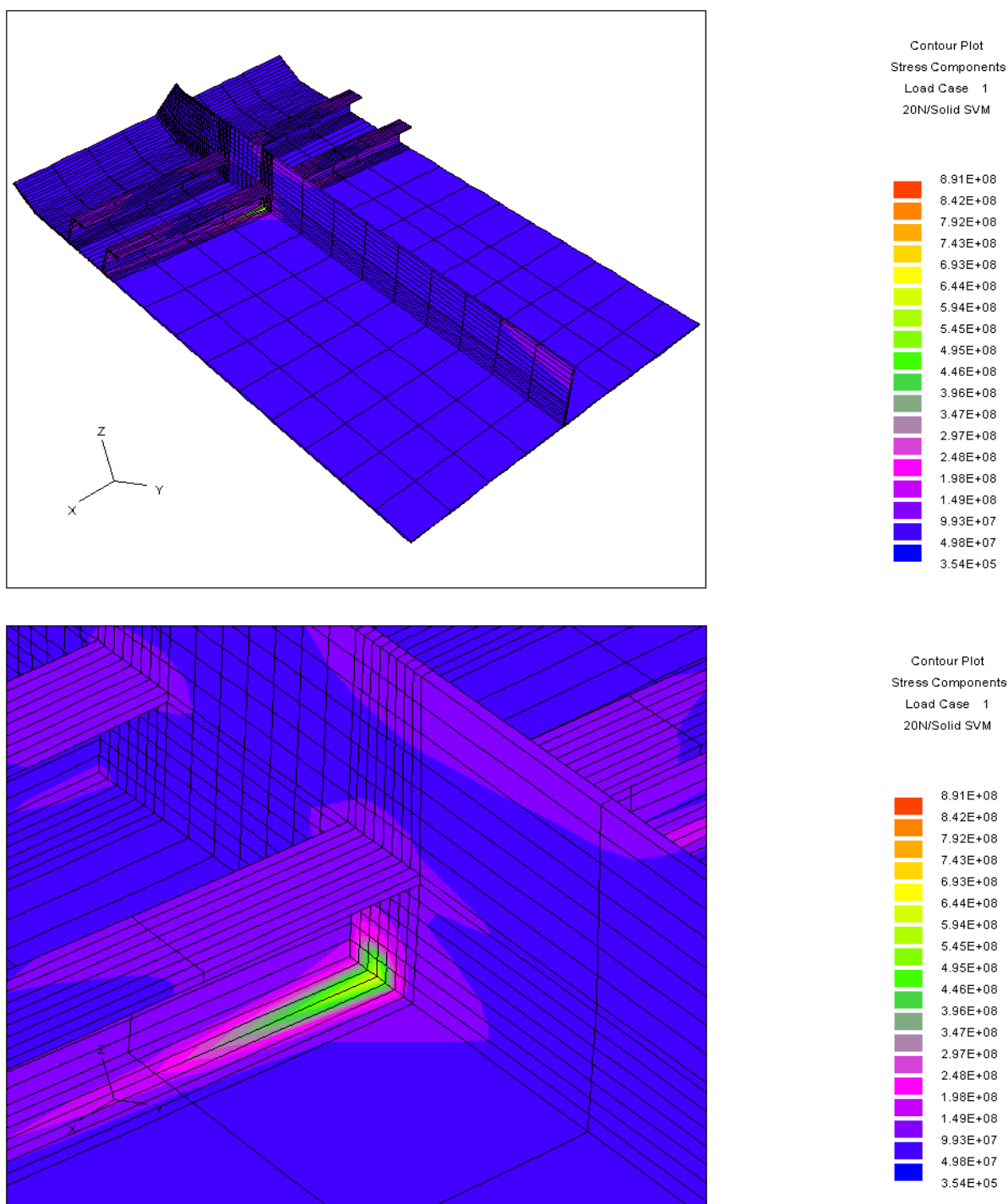




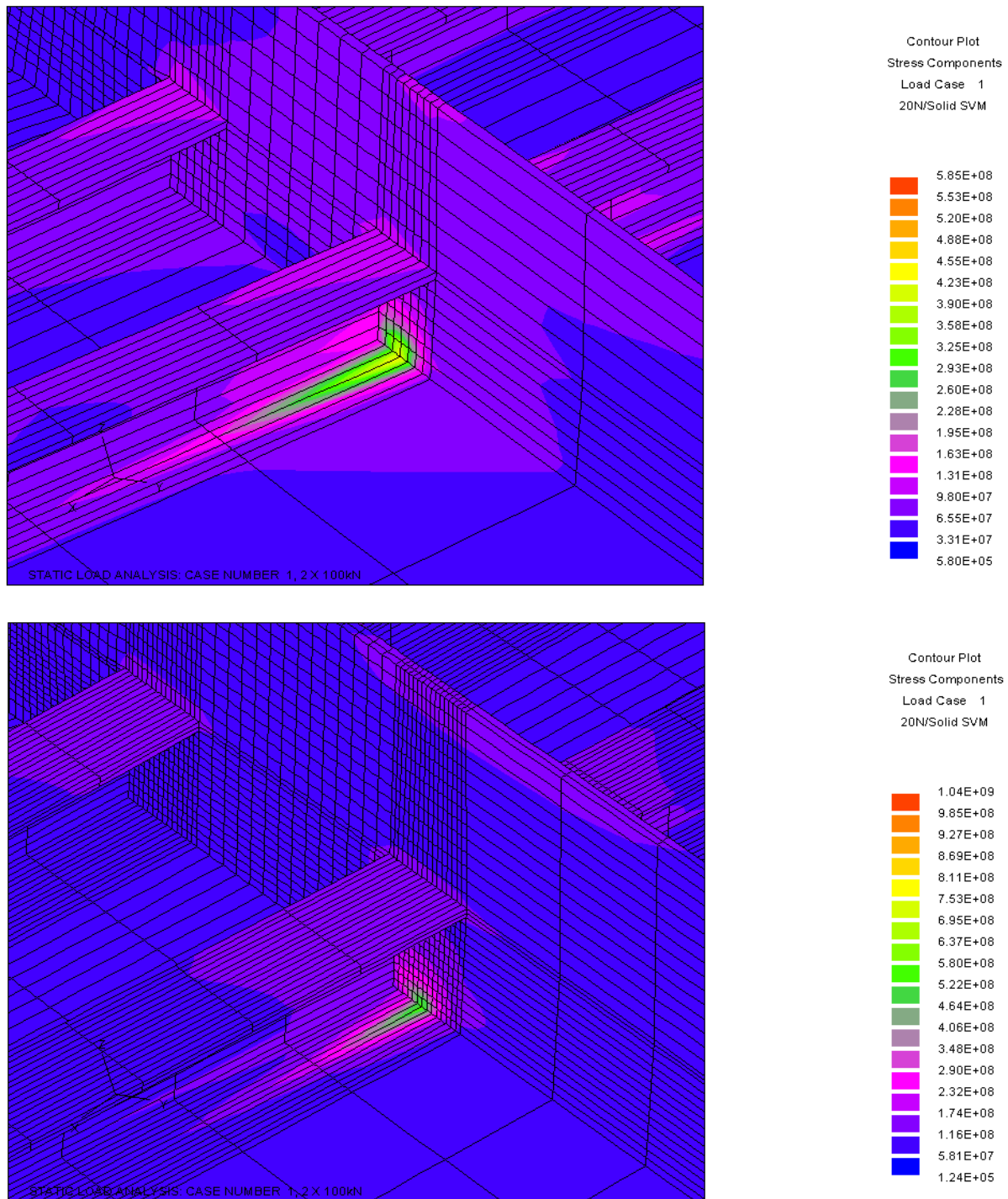
**Figure 5.5: Undeformed and Deformed Meshes of the Coarse Mesh of 20-Noded Solid Element of the Stiffened Plate Structural Component**



**Figure 5.6: Undeformed and Deformed Meshes of the Fine Mesh of 20-Noded Solid Element of the Stiffened Plate Structural Component**



**Figure 5.7: Stress Contour Predicted by the NURBS-based Solid Element Model of the Stiffened Plate Structural Component**



**Figure 5.8: Stress Contour Predicted by the Coarse (Top) and Fine (Bottom) Solid Element Models of the Stiffened Plate Structural Component**

## 6.0 CONCLUSIONS

---

In this report, we presented work performed in the second phase of a project for developing a NURBS-based computational method for the VAST finite element program. Within the initial phase of the development, a NURBS-based solid element was implemented and extensively verified for various types of linear and nonlinear analyses [1]. In the present phase, the earlier development was continued in a number of directions, including refinement of the convergence study, investigation of the accuracy problems for buckling analyses, evaluation of computational efficiency and development of a post-processing capability. All these objectives were completed successfully.

In the convergence study performed in the previous phase, the reference solution was obtained using a relatively coarse mesh of 20-noded solid elements, so the accuracy of the reference solution itself was questionable. In the present study, additional results for 20-noded solid elements were obtained using various mesh layouts and numerical integration orders. This extended convergence study confirmed that the NURBS-based solid element converges to the correct solutions with a convergence rate that is much faster than that of the traditional 20-noded isoparametric solid element.

The investigation on the accuracy of buckling analysis using the NURBS-based solid element was started from consideration of a simple test case involving a simply-supported square plate subjected to uniform bi-axial compressive stress. This test problem was selected because of the availability of analytical solutions. Numerical solutions to this problem were obtained using various NURBS-based models and different orders of numerical integrations. The numerical results were then compared with the analytical solution. These comparisons indicated that in order to obtain reliable and accurate results for linear buckling analyses, the orders of the numerical integration must be at least equal to the polynomial orders of the NURBS basis functions. This has been implemented as the default integration orders in the VAST program. Using this default rule for numerical integration, the test problem considered in the previous phase was reconsidered and very good convergence properties were obtained.

In order to establish a benchmark for computational efficiency of the NURBS-based element compared with the traditional 20-noded isoparametric solid element, a simplified structural model consisting of a hull plate and intersecting truncated bulkhead and two longitudinal stiffeners was considered. The construction of the NURBS-based model required reverse-engineering of the CAD surface model. This reverse-engineering involved the following operations: offsetting and duplicating surfaces, modifying surface definitions to ensure that the intersections between solid entities are generated properly and generating fully compatible NURBS representations at interfaces of opposite surfaces. A special-purpose computer tool was developed for this task. The NURBS-based model of the stiffened plate structural component was used in a linear static analysis and the results were compared with two 20-noded solid element models with different levels of mesh refinement. The benchmark results indicated that in order to achieve comparable accuracy, the NURBS-based element requires a much coarser mesh, leading to a significant saving of computation time.

The results from the present study further confirmed the potential of the NURBS-based solid element in engineering analyses and also the study significantly improved its readiness for practical applications.

## **7.0 REFERENCES**

---

- [1] L. Jiang, "Development of a NURBS-based Computational Method for the VAST Finite Element Program", Martec Limited, DRDC Atlantic Contractor Report CR 2007-187, July 2007.
- [2] S. Timoshenko, Theory of Elastic Stability, McGraw-Hill, New York, 1936.

This page intentionally left blank



## **Distribution list**

---

DRDC Atlantic CR 2008-118

### **Internal distribution**

4 Author (2 hard copies, 2 CDs)

5 DRDC Atlantic Library (1 hard copy, 4 CD's)

**Total internal copies: 9**

### **External distribution (within Canada by DRDKIM)**

1 Library and Archives Canada

1 NDHQ/DRDKIM 3

1 NDHQ/DMSS 2  
Louis St. Laurent Bldg.  
555 boul de la Carriere  
Ottawa, Hull K1A OK2

1 Canadian Forces Maritime Warfare School  
Attention: Commanding Officer  
PO Box 99000 STN Forces  
Halifax, NS B3K 5X5

**Total external copies: 4**

**Total copies: 13**

This page intentionally left blank

DOCUMENT CONTROL DATA		
(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)		
1. ORIGINATOR (the name and address of the organization preparing the document.. Organizations for whom the document was prepared, e.g. Centre sponsoring a contractor's report, or tasking agency, are entered in section 8.)  <b>MARTEC Limited, 1888 Brunswick Street, Suite 400, Halifax, Nova Scotia, Canada B3J 3J8</b>		2. SECURITY CLASSIFICATION (overall security classification of the document including special warning terms if applicable).  <b>UNCLASSIFIED</b>
3. TITLE (the complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S, C, R or U) in parentheses after the title).  <b>Development of a NURBS-Based Computational Method for the VAST Finite Element Program: Phase 2 – Final Report</b>		
4. AUTHORS (Last name, first name, middle initial. If military, show rank, e.g. Doe, Maj. John E.)  <b>Jiang L. and Teng E.</b>		
5. DATE OF PUBLICATION (month and year of publication of document)  <b>October 2008</b>	6a. NO. OF PAGES (total containing information Include Annexes, Appendices, etc).  <b>58</b>	6b. NO. OF REFS (total cited in document)  <b>2</b>
7. DESCRIPTIVE NOTES (the category of the document, e.g. technical report, technical note or memorandum. If appropriate, enter the type of report, e.g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered).  <b>CONTRACT REPORT</b>		
8. SPONSORING ACTIVITY (the name of the department project office or laboratory sponsoring the research and development. Include address). <b>Defence R&amp;D Canada - Atlantic PO Box 1012 Dartmouth, NS, Canada B2Y 3Z7</b>		
9a. PROJECT OR GRANT NO. (if appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant).  <b>11 GT 3</b>	9b. CONTRACT NO. (if appropriate, the applicable number under which the document was written).  <b>W7707-06-3597/001/HAL Task 09</b>	
10a ORIGINATOR'S DOCUMENT NUMBER (the official document number by which the document is identified by the originating activity. This number must be unique to this document.)  <b>Martec Technical Report # TR-08-09 r1</b>	10b OTHER DOCUMENT NOs. (Any other numbers which may be assigned this document either by the originator or by the sponsor.)  <b>DRDC Atlantic CR 2008-118</b>	
11. DOCUMENT AVAILABILITY (any limitations on further dissemination of the document, other than those imposed by security classification)  ( <input checked="" type="checkbox"/> ) Unlimited distribution (    ) Defence departments and defence contractors; further distribution only as approved (    ) Defence departments and Canadian defence contractors; further distribution only as approved (    ) Government departments and agencies; further distribution only as approved (    ) Defence departments; further distribution only as approved (    ) Other (please specify):		
12. DOCUMENT ANNOUNCEMENT (any limitation to the bibliographic announcement of this document. This will normally correspond to the Document Availability (11). However, where further distribution (beyond the audience specified in (11) is possible, a wider announcement audience may be selected).  <b>UNLIMITED</b>		

13. **ABSTRACT** (a brief and factual summary of the document. It may also appear elsewhere in the body of the document itself. It is highly desirable that the abstract of classified documents be unclassified. Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U). It is not necessary to include here abstracts in both official languages unless the text is bilingual).

This report presents a recent project for further verification and enhancement of a recently developed NURBS-based computational method in VAST. This method has the potential for close integration of engineering analysis capability with Computer Aided Design (CAD) systems. It also has the advantages of preserving exact geometry of structures to be analyzed and providing a convenient means for mesh refinement. In a previous project, the NURBS-based formulation was implemented into the VAST program and verified for a wide variety of engineering analyses, including linear static, eigenvalue and nonlinear analyses involving plastic post-collapse responses of thin shell structures. In this work, additional verification of the NURBS-based element is performed and the numerical problems observed in the previous study were also investigated and resolved. A post-processing capability for the NURBS-based element is developed and the possibility for applying this element for solving practical engineering problems is also explored. This study confirmed advantages of NURBS-based elements from the computational point of view. An automatic model generator for NURBS-based elements still needs to be developed in the future.

14. **KEYWORDS, DESCRIPTORS or IDENTIFIERS** (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document. They should be selected so that no security classification is required. Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included. If possible keywords should be selected from a published thesaurus. e.g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified. If it not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title).

Finite element analysis  
Structural analysis  
Non-Uniform Rational B-Splines  
NURBS  
Computer Aided Design  
CAD  
Linear  
Non-Linear  
Buckling

This page intentionally left blank.

## **Defence R&D Canada**

Canada's leader in defence  
and National Security  
Science and Technology

## **R & D pour la défense Canada**

Chef de file au Canada en matière  
de science et de technologie pour  
la défense et la sécurité nationale



[www.drdc-rddc.gc.ca](http://www.drdc-rddc.gc.ca)